Multiframe Steel Codes

Windows Version 16

User Manual

© Bentley Systems, Incorporated 2013

License & Copyright

Multiframe Steel Codes software & User Manual
© 2013 Bentley Systems, Incorporated

Table of Contents

License & Copyright iii
Table of Contents v
About this manual 1
Chapter 1 Getting Started 3
About Multiframe Steel Codes 3
Design Codes 3
Installing Multiframe Steel Codes 4
Starting Multiframe Steel Codes 4
Adding or Removing Steel Design Codes 4
Design Overview 4
Design Members 4
Bending Checks 4
Tension Checks 4
Compression Checks 5
Combined Checks 5
Serviceability Checks 5
Seismic Checks 5
Checking a member 5
Designing a member 5
Reporting 5
Windows 6
Frame Window 6
Data Window 6
Result Window 6
Plot Window 6
Report Window 7
Design Members 7
Viewing Results Using Design Members 8
Design Member Symbols 8
Rendering Design Members 9
Coordinate Systems 9
Properties for Design 9
Shear Area 10
Chapter 2 Using Multiframe Steel Codes 11
Design Procedure 11
Working with Design Members 12
Setting Design Properties 12
Setting Design Properties 13
Setting Section Type 15
Setting Steel Grade 15
Setting Design Constraints 18
Setting Section Constraints 18
Setting Frame Type 19
Setting Allowable Stresses 19
Setting Acceptance Ratio 20
Setting Capacity Factors 20
Checking a Frame 20
Displaying Efficiency 22
Governing Load Cases 22
Designing a Frame 23
Optimum Sections 24
Tips On Optimisation 25

Finding Design Values 25
Printing 25
Printing the Report Window 26
Saving your Work 26
Saving the report 26
Chapter 3 ASD and AIJ 27
Design Checks - ASD and AIJ 27
Bending - ASD and AIJ 27
Design Constraints (AIJ) 27
Unbraced Length - ASD and AIJ 27
Bending Coefficient (ASD) 28
Web Stiffener Spacing - ASD and AIJ 28
Bending Dialog - ASD and AIJ 28
Tension - ASD and AIJ 28
Bolt Holes - ASD and AIJ 29
Area Reduction - ASD and AIJ 29
Tension Dialog - ASD and AIJ 29
Compression - ASD and AIJ 29
Compression Dialog - ASD and AIJ 30
Combined Actions - ASD and AIJ 31
Default Design Properties - ASD and AIJ 31
Code Clauses Checked - ASD and AIJ 32
ASD Clauses Checked 32
AIJ Clauses Checked 33
Short Term Loads for AIJ 34
Chapter 4 AS4100 and NZS3404 35
Notation - AS4100 and NZS3404 35
Design Checks - AS4100 and NZS3404 35
Bending - AS4100 and NZS3404 35
Lateral Restraints - AS4100 and NZS3404 36
Unbraced Length (l_e) and Bending Coefficient (α_m) - AS4100 and NZS3404 37
Web Stiffener Spacing - AS4100 and NZS3404 37
Load Height - AS4100 and NZS3404 37
Bending Dialog - AS4100 and NZS3404 37
Generate Lateral Restraints Dialog - AS4100 and NZS3404 39
Tension - AS4100 and NZS3404 39
Bolt Holes - AS4100 and NZS3404 40
Correction Factor - AS4100 and NZS3404 40
Tension Dialog - AS4100 and NZS3404 40
Compression - AS4100 and NZS3404 41
Unbraced Length - AS4100 and NZS3404 41
Compression Dialog - AS4100 and NZS3404 41
Combined Actions - AS4100 and NZS3404 42
Serviceability - AS4100 and NZS3404 42
Serviceability Dialog - AS4100 and NZS3404 42
Seismic (NZS3404) 43
NZS3404 Seismic Dialog 43
Default Design Properties - AS4100 and NZS3404 44
Code Clauses Checked - AS4100 and NZS3404 45
AS4100 Clauses Checked 45
NZS3404 Clauses Checked 46
Chapter 5 LRFD 49
Notation - LFRD 49

Design Checks - LFRD 49
Bending - LFRD 49
Lateral Restraints - LFRD 50
Unbraced Length (L_b) and Bending Coefficient (C_b) - LFRD 51
Web Stiffener Spacing - LFRD 51
Bending Dialog - LFRD 51
Generate Lateral Restraints Dialog - LFRD 52
Tension - LFRD 53
Bolt Holes - LFRD 53
Reduction Coefficient - LFRD 54
Tension Dialog - LFRD 54
Compression - LFRD 54
Compression Dialog - LFRD 55
Combined Actions - LFRD 56
Serviceability - LFRD 56
Serviceability Dialog - LFRD 56
Default Design Properties - LFRD 57
Code Clauses Checked - LFRD 58
LRFD Clauses Checked 58
LRFD SAM Clauses Checked 59
Chapter 6 BS5950 61
Notation - BS5950 61
Design Checks - BS5950 61
Bending - BS5950 62
Lateral and Torsional Restraints - BS5950 63
Unbraced Length (L_b) and Bending Coefficient (m_{LT}) - BS5950 63
Web Stiffener Spacing - BS5950 64
Load Height - BS5950 64
Bending Dialog - BS5950 64
Generate Lateral Restraints Dialog - BS5950 65
Tension - BS5950 66
Bolt Holes - BS5950 66
Area Reduction Coefficient - BS5950 67
Tension Dialog - BS5950 67
Compression - BS5950 68
Unbraced Lengths and Effective Length Factors - BS5950 69
Column Segments - BS5950 69
Compression Dialog - BS5950 69
Combined Actions - BS5950 71
Serviceability - BS5950 72
Serviceability Dialog - BS5950 72
Default Design Properties - BS5950 72
Code Clauses Checked - BS5950 73
Chapter 7 AS/NZS4600 77
Setting Properties - AS/NZS4600 77
Bending - AS/NZS4600 79
Tension - AS/NZS4600 82
Compression - AS/NZS4600 83
Unbraced Length - AS/NZS4600 83
Combined Actions - AS/NZS4600 84
Design Properties - AS/NZS4600 84
Steel Grade - AS/NZS4600 85
Code Checks - AS/NZS4600 86
Design Checking Procedure 86

References - AS/NZS4600 86
Chapter 8 AISI 89
Setting Properties - AISI 89
Bending - AISI 91
Tension - AISI 94
Compression - AISI 95
Unbraced Length - AISI 95
Combined Actions - AISI 96
Design Properties - AISI 96
Steel Grade - AISI 97
Code Checks - AISI 98
Design Checking Procedure 98
References - AISI 98
Chapter 9 AISC 2005/2010 101
Notation – AISC 2005/2010 101
Design Checks - AISC 2005/2010 102
Bending - AISC 2005/2010 102
Lateral Restraints - AISC 2005/2010 102
Unbraced Length (L_b) - AISC 2005/2010 103
Web Stiffener Spacing - AISC 2005/2010 103
Bending Dialog AISC 2005/2010 103
Generate Lateral Restraints Dialog - AISC 2005/2010 105
Tension - AISC 2005/2010 105
Bolt Holes - AISC 2005/2010 106
Shear Lag Factor - AISC 2005/2010 106
Tension Dialog - AISC 2005/2010 106
Compression - AISC 2005/2010 107
Unbraced Length - AISC 2005/2010 107
Compression Dialog – AISC 2005/2010 107
Combined Actions – AISC 2005/2010 110
Serviceability - AISC 2005/2010 110
Serviceability Dialog – AISC 2005/2010 110
Default Design Properties – AISC 2005/2010 110
Code Clauses Checked – AISC 2005/2010 111
Chapter 10 Eurocode 3 115
Notation – Eurocode 3 115
Design Checks - Eurocode 3 115
Bending - Eurocode 3 115
Lateral Restraints - Eurocode 3 116
Unbraced Length (L_b) - Eurocode 3 117
Web Stiffener Spacing - Eurocode 3 117
Bending Dialog Eurocode 3 117
Generate Lateral Restraints Dialog - Eurocode 3 118
Tension - Eurocode 3 119
Bolt Holes - Eurocode 3 119
Tension Dialog - Eurocode 3 119
Compression - Eurocode 3 120
Unbraced Length - Eurocode 3 120
Compression Dialog – Eurocode 3 121
Serviceability - Eurocode 3 123
Serviceability Dialog - Eurocode 3 123
National Annex 123
National Annex Dialog – Eurocode 3 123
Default Design Properties - Eurocode 3 124

Code Clauses Checked – Eurocode 3 125
Chapter 11 User Code 127
User Codes - Concepts 127
User Code – Procedures 127
Chapter 12 Multiframe Steel Codes Reference 131
Windows 131
Frame Window 131
Data Window 131
Load Window 131
Result Window 131
Plot Window 131
Report Window 132
Menus 132
Group Menu 132
Design Menu 132
Code Submenu 134
Display Menu 135
Efficiency Submenu 135
Help Menu 138
References 140
Index 141

About this manual

This manual is about Multiframe Steel Codes, a structural steel design application for the Windows operating system. Multiframe Steel Codes is an add-on module to the Multiframe structural analysis software.

Chapter 1
; provides an overview of Multiframe Steel Codes and it's capabilities. Once you are familiar with the basic concepts and knowledge required to use Multiframe Steel Codes, you may refer to the detailed instructions in Chapter two.

Chapter 2
Using Multiframe Steel Codes; gives step-by-step instructions of how to use Multiframe Steel Codes. It describes all the commands and functionality provided by Multiframe Steel Codes except for the details specific to each of the design codes. The following chapters provide the information particular to each design codes supported by Multiframe Steel Codes.

Chapter 3
ASD and AIJ; describes the design checks, dialogs and design properties specific to the American ASD and Japanese AIJ allowable stress steel design codes.

Chapter 4
AS4100 and NZS3404; the design checks, capabilities and limitations, dialogs and design properties specific to the Australian AS4100 and New Zealand NZS3404 limit state steel design codes.

Chapter 5
LRFD; describes the design checks, capabilities and limitations, dialogs and design properties specific to the American LRFD limit state steel design code.

Chapter 6
BS5950; describes the design checks, capabilities and limitations, dialogs and design properties specific to the British BS5950 limit state steel design code.

Chapter 7
AS/NZS4600; describes the design checks, capabilities and limitations, dialogs and design properties specific to the AS/NZS4600 steel design code.

Chapter 8
AISI; describes how the user can specify an alternative set of design rules that can be used by Multiframe Steel Codes when designing a frame.

Chapter 9
AISC 2005; describes the design checks, capabilities and limitations, dialogs and design properties specific to the AISC 2005 LRFD and ASD steel design codes.

Chapter 10
Eurocode 3, describes the design checks, capabilities and limitations, dialogs and design properties specific to the Eurocode 3 steel design code.

Chapter 11
User Code; explains how to enter custom design rules.

Chapter 12
Multiframe Steel Codes Reference, describes gives an overview of the windows and menus of Multiframe Steel Codes and a summary of the commands used.

Chapter 1
Getting Started

This chapter provides an introduction to Multiframe Steel Codes. It outlines the basic concepts and knowledge needed to use the program as well as the additional functionality it introduces to the Multiframe user interface in the following sections:

- About Multiframe Steel Codes
- Design Codes
- Installing Multiframe Steel Codes
- Design Overview
- Windows
- Design Members
- Coordinate Systems
- Properties for Design
- Shear Area

About Multiframe Steel Codes

Multiframe Steel Codes is an add-in module for Multiframe that is used for checking or designing a steel frame in accordance with various codes of practice. After analysing a frame in Multiframe you can use Multiframe Steel Codes to check the members in the structure for compliance with a design code. You can also use Multiframe Steel Codes to choose the lightest weight sections, which satisfy the design criteria.

A word of caution:

Multiframe Steel Codes is a very useful aid to the design of steel structures. It is NOT an automatic design tool and it should be used in conjunction with professional engineering judgment to produce well-designed frames.

Design Codes

Multiframe Steel Codes supports checking and designing of your structure in accordance with a range of design codes. At present, Multiframe Steel Codes allows you to use

- **AIJ (Architectural Institute of Japan 1979)**
- **ASD (American Institute of Steel Construction Allowable Stress Design, 9th Ed 1989)**
- **AS4100 (Australian Steel Design Code, Standards Australia, 1990)**
- **LRFD (American Institute of Steel Construction Load and Resistance Factor Design, December 27th 1999)**
- **NZS 3404 (New Zealand Steel Design Code, Standards New Zealand, 1997)**
- **BS5950 (British Steel Design Code, British Standards Institution, 2000)**
- **AS/NZS4600 (Australian/New Zealand Steel Design Code, Australian Standards Institution, 2005)**
- **AISI (North American Specification for the Design of Cold-formed Steel Structural Members ", AISI Standards, 2001 Edition)**
- **A user definable allowable stress code**

Other design codes will be supported in future releases of Multiframe Steel Codes.

Only design codes licensed by the user will be active in the Code menu. A detailed description of the design checks performed by Multiframe Steel Codes for each of the design codes is given in the following Chapters.

Installing Multiframe Steel Codes

Multiframe Steel Codes is installed as part of the Multiframe Suite installer. For instructions, please see: http://www.formsys.com/installation or the installation guide on the installation CD.

Starting Multiframe Steel Codes

Because Multiframe Steel Codes is an add-on to the Multiframe application and runs fully within the Multiframe application, you can not start Multiframe Steel Codes separately. After installing the required Multiframe Steel Codes code and starting the Multiframe application, you will see additional menu items appear. If this is not the case, you have to manually enable the Multiframe Steel Codes licenses from the Licensing tab from the Edit | Preferences dialog in Multiframe. Only installed design codes can be selected, others will be greyed out.

Adding or Removing Steel Design Codes

If you wish to add or remove Steel Design codes, you should run the original installer again and select Modify. See the Installation Guide, section Repairing or Modifying the installation for more information.

Design Overview

Multiframe Steel Codes is used to check the compliance of a member or design a member to a specific steel design code. Each of the steel design codes supported by Multiframe Steel Codes is divided into a number of design checks. The user can specify which of these checks are performed when a member is designed or checked. The design checks are grouped into the categories; Bending, Tension, Compression, Combined, and Seismic. However, not all codes have checks in each category and the design checks listed within each category vary according to the design code performed when a member is designed or checked.

Design Members

A design member is a single member or a group of co-linear members that are to be considered as a single member for the purposes of design. In this manual, the term *member* often refers to a design member when used in the context of design.

Bending Checks

Bending checks are usually used on members which resist the applied loads by flexural and shear actions. Typically the horizontal members in a frame will support the live and gravity loads in this way. A member may be subject to flexure and shear in either the major or minor axis directions (or both) depending the orientation of the section and the direction of the loading.

Tension Checks

Tension checks are performed on members that are subject to axial tension. This would include members such as bracing and members in trusses which are under tension.

Compression Checks

Compression checks are used on members that support axial compression. Columns and bracing in frames and compression members in trusses are some of the types of members that are likely to be checked using this option. Some codes may also include a check on the slenderness of a member.

Combined Checks

When a member is subject to combined actions, generally bi-axial bending or a combination of axial tension or compression and bending, it is likely to be necessary to carry out a combined check on the member's performance.

Serviceability Checks

Serviceability checks allow the user to specify the maximum deflection of a member. For some codes the serviceability checks have been included with the Bending checks.

Seismic Checks

When a structure is located in a seismic region some additional design requirements are imposed by some design codes. This typically requires that certain members within a steel frame be designed for ductility.

Checking a member

Multiframe Steel Codes can be used to check the compliance of a member to a steel design code. When checking a member, Multiframe Steel Codes computes an efficiency for each of the active design checks. The efficiency is a measure of the member's design action, design stress or deflection expressed as a percentage of the allowable capacity as calculated using the design rules. That is, an ideal member is loaded or stressed to 100% of its allowable design capacity (or slightly less) and a member labelled as being 50% efficient is twice as strong as it needs to be.

When checking a member, the user has the option to output the design calculations performed by Multiframe Steel Codes to the report window.

Designing a member

As well as helping to check a frame's compliance with the design rules, Multiframe Steel Codes can also help you to select the lightest weight section that satisfies the design rules. In this case, Multiframe Steel Codes iterates through the current group of sections until it finds the optimal section that satisfies the selected design checks. Multiframe Steel Codes also computes the efficiency of the optimal section for each of the active design checks.

Reporting

Multiframe Steel Codes can produce a detailed report of the design calculations it performs for each member. The level of reporting can be tailored by the user to reduce the amount of detail shown in the report. The design calculations produced by Multiframe Steel Codes are displayed in the Report Window. You can copy and paste from this window into other programs, save from it in RTF format, or directly print the contents of the window.

Alternatively you can choose to output the design calculations directly to Microsoft Word. This option can be specified in the Preferences Dialog. If this option is selected and Microsoft Word is installed on the computer, Multiframe will automatically run Word when it is required for reporting. The design report will be placed into a new document in Word. This method of reporting is very fast and gives you direct access to the advanced printing and formatting options of Microsoft Word.

Windows

When Multiframe Steel Codes is activated within Multiframe the content and/or the behaviour of the Frame, Plot, Data and Results windows is extended and the Report window is used to display a summary of the design checks made by Multiframe Steel Codes. You can also paste text and graphics into the report to help document your calculations.

The following sections document the additional content and behaviour of the windows in Multiframe when Multiframe Steel Codes is activated.

Frame Window

When using Multiframe Steel Codes, the Frame window sets up the design properties for the members in the frame. You can do this by selecting members and then using the items in the Design menu to set the various design values. You can also change the design properties of a member by double clicking on it in the Frame window. This will produce an extended Member Properties dialog that contains separate tabs for setting many of the design options. The same dialog appears if you choose Design Details from the Design menu.

Data Window

The Data window includes an additional table named Design Details. You can display this table by choosing Design Details from the Data sub-menu under the Display menu.

This table displays all of the design information required for each member so that Multiframe Steel Codes can carry out the design checks. You can change this data by clicking on the value you wish to change, typing in the new value, and typing Enter. You may also copy and paste data to and from the table.

Numbers in this table that are displayed in Italics (in the Cb, Cmx and Cmy columns) will be calculated by Multiframe Steel Codes, you do not have to enter them. If you wish however, you can override the calculation of these values by typing in a value to be used. Any values you enter will be displayed in normal type. To revert to the automatic calculation of any value, type in a value of zero.

Result Window

In addition to the tables of results displayed in Multiframe, the Result Window contains an additional table named Design Efficiency. If a member was checked for its compliance to a code then this table displays the efficiency for each design check. If Multiframe Steel Codes was used to find the optimal section size then the table displays the optimal section as well as the efficiency of that section.

Plot Window

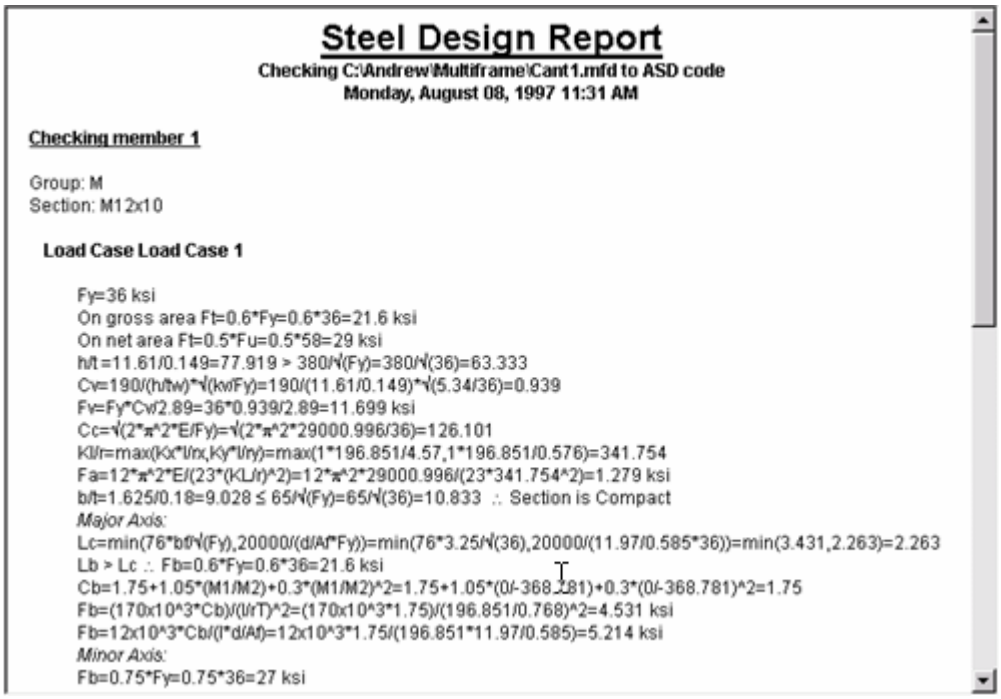
With Multiframe Steel Codes there is an additional display function in the Plot window that lets you display a graphical representation of the efficiency of the members relative to the design code requirements.

You can display efficiency by choosing the required item from the Efficiency sub-menu under the Display menu. This displays the same information that is displayed numerically in the Efficiency table in the Result window. Multiframe Steel Codes uses a colour display to show the stress or deflection level in the member relative to its allowable value. The scale on the right hand side of the window indicates the relationship between the colours and the level of efficiency. Members that are more highly loaded, stressed or deflected than the level allowed by the code are shown in red.

You can use the Symbols command from the Display menu to turn on the display of Plot values. When this option is on, the values of the efficiency will also be displayed on each member that has been checked.

Report Window

This window is used to create a progressive summary of the design that has been carried out.



This report can be edited via Cut, Copy, Paste and Clear, printed, or saved to and recalled from a disk file. You can type directly into the report or edit the text in the report however modifying the properties of the fonts in equations can easily corrupt the formatting of the design equations as the Greek characters and mathematical symbols are displayed using the Symbol font.

Design Members

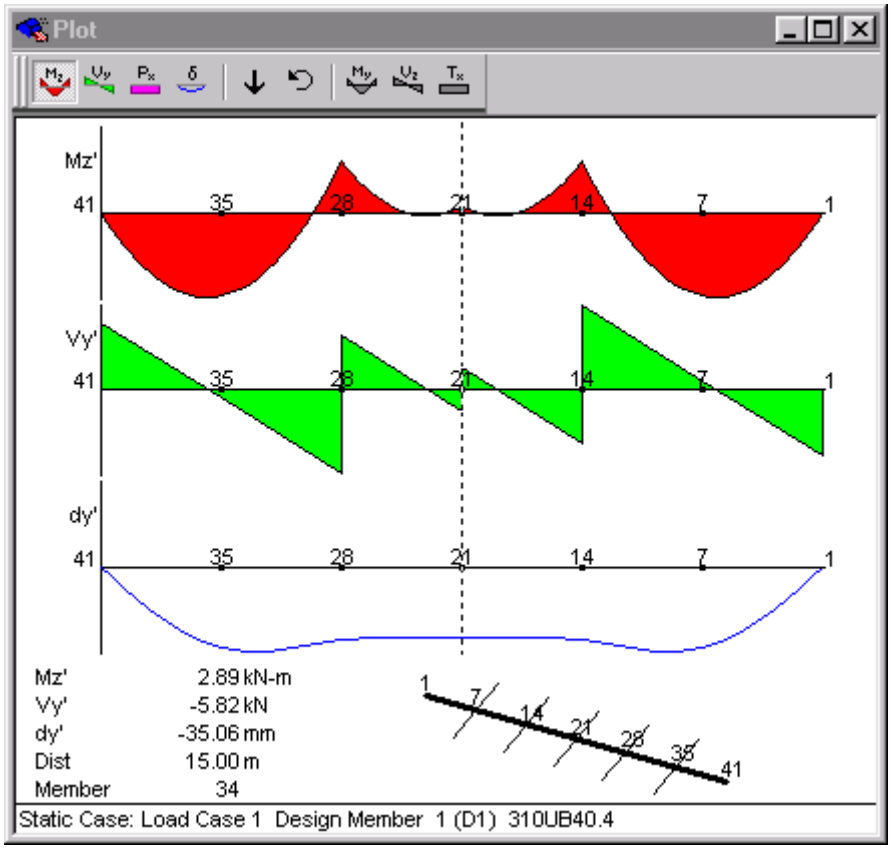
A design member is a single member, or a series of connected members that can be considered as a single member for design purposes. By default, each member in the frame is a design member.

Members to be grouped together into a Design Member must satisfy the following conditions-

- **All members must have the same section type**
- **All members must have the same orientation**
- **All members must be rigidly connected internally (ends may be released)**
- **All members must be approximately co-linear**
- **All members must be connected with the local x' axis facing the same direction**
- **Members may have rigid offsets at internal joints but the flexible portions of the members must be continuous within the design group.**
- **There must not be any restraints on the internal connecting nodes**

Viewing Results Using Design Members

The action and displacement diagrams for a design member may be viewed in the Plot Window. Double-clicking on a design member produces a local member diagram for the entire design member. If the design member consists of more than one member, the diagram for a single member can be examined by simply clicking on that member within the diagram.



Design Member Symbols

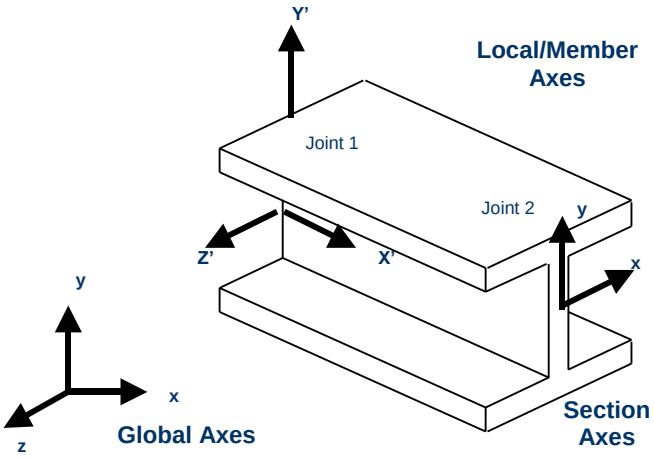
In the Symbols dialog there are three check boxes grouped together which are dedicated to viewing design members. If Design Members is checked then design members containing more than a single member are displayed in the Frame window by a patterned blue overlay. If Labels is checked the labels of the design members are displayed in all the drawing windows. If Numbers is checked the numbers of all the design members used in design are displayed in all the drawing windows.

Rendering Design Members

Design members are rendered in the Frame and Load windows as a single member.

Coordinate Systems

Much of the design information and many of the design variables are described relative to the major and minor axes of the section used for each member. This corresponds to the same terminology used to describe the properties of a section e.g. Ixx for moment of inertia about the major (or strong) axis and Iyy about the minor (or weak) axis.



The coordinate systems corresponding to the naming conventions for the various results of analysis, section properties and design values are shown in the diagram above. Structure coordinates and global loads are defined relative to the Global Axes, member actions, deflections and stresses resulting from the Multiframe analysis are defined relative to the local member axes and design values are defined relative to the section axes. Whenever a design variable carries a subscript this indicates that it applies to the corresponding section axis. (E.g. fbx refers to the design bending stress about the x-axis)

Properties for Design

When checking or designing structures, Multiframe Steel Codes uses sections properties stored in the Sections Library. The key properties used by Multiframe Steel Codes are:

Property
Cross sectional area
Major moment of inertia
Minor moment of inertia
Young's Modulus
Depth
Breadth or Width
Flange thickness
Web thickness

Major radius of gyration

Minor radius of gyration

Radius of gyration about weakest axis

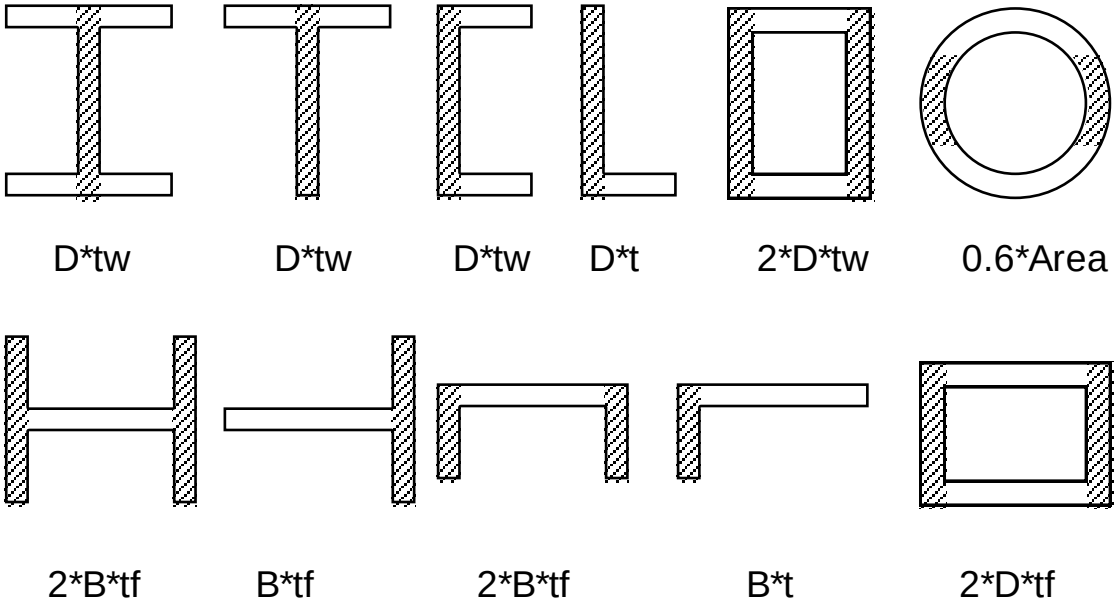
Plastic modulus about major axis

Plastic modulus about minor axis

When you add a section to the Sections Library you must ensure that all of the properties above are correctly entered and are all non-zero.

Shear Area

When calculating shear stresses for comparison with allowable shear stresses, Multiframe Steel Codes uses the following shear areas or the full sectional area for other sectional shapes.



Chapter 2
Using Multiframe Steel Codes

This chapter describes how to use Multiframe Steel Codes with step-by-step instructions on the basics of using the program in the following sections:

- Design Procedure
- Working with Design Members
- Setting Design Properties
- Setting Design Properties
- Setting Section Type
- Setting Steel Grade
- Setting Design Constraints
- Setting Section Constraints
- Setting Frame Type
- Setting Allowable Stresses
- Setting Acceptance Ratio
- Setting Capacity Factors
- Checking a Frame
- Designing a Frame
- Printing
- Saving your Work
- Saving the report

Design Procedure

The basic procedure for checking or designing a frame using Multiframe Steel Codes is as follows;

- **Set up the structure and loading**
- **Carry out the analysis**
- **Check the results to ensure your structural model is correct**
- **If necessary, group members into design members**
- **Enter the design information (such as effective lengths, steel grades etc.)**
- **Carry out the design checks or search for the optimum sections**

When you use the Check or Design commands you have the option of specifying which design checks will be carried out. The types of checks are grouped into the categories; Bending, Tension, Compression, Combined, Serviceability (AS4600 and NZS3404 only) and Seismic (NZS3404 only). The design checks listed within each category vary according to the design code. The user may specify which of these checks are performed when a member is designed or checked using Multiframe Steel Codes.

Working with Design Members

When designing a frame it is often convenient to group members together and treat them as a single member for the purposes of design. This is often the case when a physical member in a frame has been subdivided into a number of members in the Multiframe model.

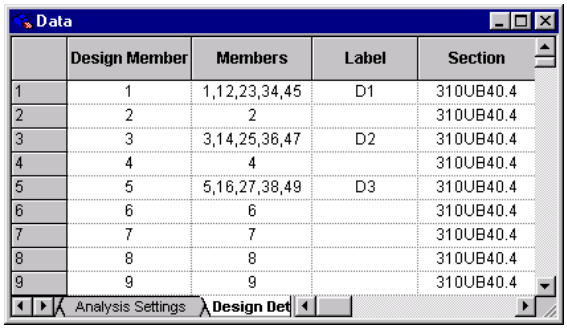
Members can be combined into a single design member in the Frame Window. To create a design member,

- **Select the members to be grouped**
- **Choose "Create Design Members" from Group menu.**

or

- **Press Ctrl+D**

The members that form each design member are displayed in the Design Details and Design Efficiency data tables.



To delete or split design members, select members that are part of the design member(s) and choose "Ungroup Members" from the Design menu.

Setting Design Properties

Before doing the checks, it is necessary to enter basic design data such as effective length, grade of steel etc. This information can either be entered in the Frame, Load or Plot windows by selecting design members and using the commands under the Design menu, or it can be entered in tabular form in the Data window. The actual design parameters that can be changed by the user will vary according to the current design code. A list of design variables and their default values are described in subsequent chapters in this manual.

Although most of the design variables are pre-set to the most commonly used values, you will probably want to enter the design information for at least some of the members in the frame that you wish to check. You set design variables by selecting the members you wish to change and then choosing the appropriate command from the Design menu.

It is not necessary to enter the design data for all of the design checks. Usually you will want to check some members for bending, others for compression and so on. The items under the Design menu help you enter just the required information depending on what type of check you are doing. The design properties are grouped according the categories described above and the items in the Design menu reflect these groupings. The dialogs displayed by each of these commands will vary according the current design code.

Bending

When performing a bending check, you may need to specify a number of properties relating to the unbraced length, location and type of lateral restraints, and the stiffener spacing on the member.

Tension

Tension checks usually require the user to specify the area of holes in the cross section and a coefficient to account for the distribution of end forces or used to computing effective net area of the section.

Compression

When checking or designing members for compression, it is necessary to specify the effective length and unbraced length of the member.

Combined Actions

Some design codes require the user to specify a coefficient that accounts for the distribution of moments along a member.

Serviceability

With some design codes, it may be necessary to specify the deflection limits used in checking the serviceability of a member.

Seismic

Some design codes require a member to be categorised according to the required ductility of the member.

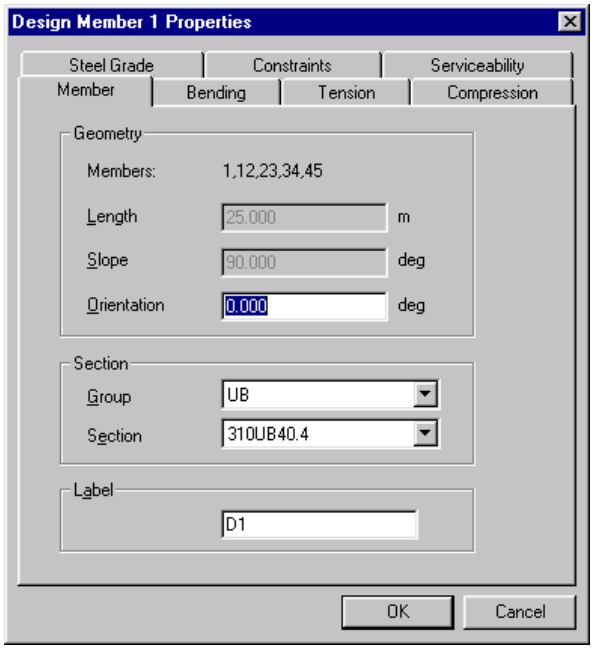
For some design codes, no design data is required for the design checks in a particular category and so the menu item will not be enabled. In other codes, there are no design checks performed within a particular category and the menu item will again be disabled.

Setting Design Properties

Sometimes you may wish to set or review all of the design properties for a member at once. This may be quicker than setting each of the design values in turn using the commands above.

To set all of the design variables

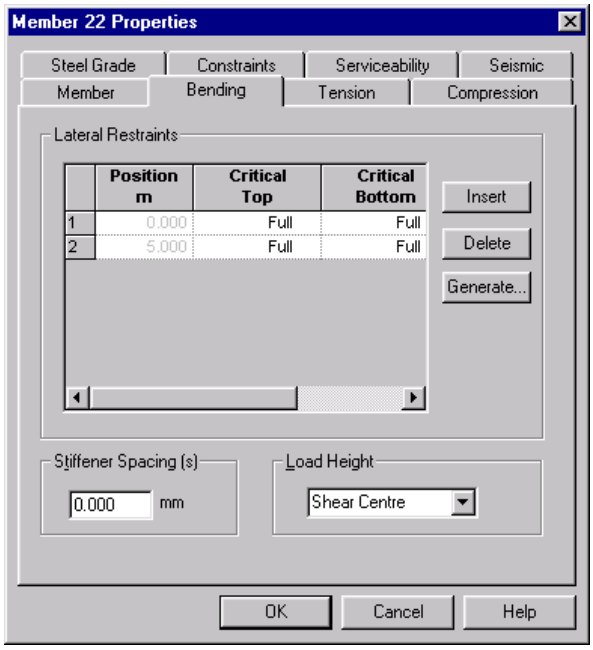
- **Select the required members in the Frame window**
- **Choose Design Details from the Design menu**



AS4100 shown

- **Enter the design values**
- **Click OK**

As a short cut, you can examine and change the design details for a single member by double clicking on it in the Frame window.



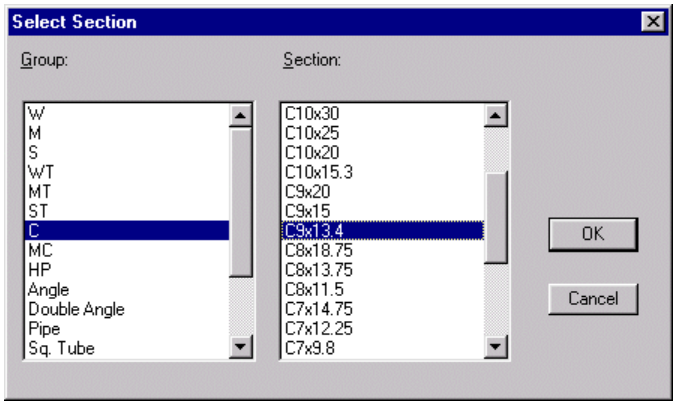
NZS3404 shown

Setting Section Type

If necessary you can change the section type of a member manually in Multiframe Steel Codes. Note however, that if you do so, you will need to re-analyse the structure using the Analyse command from the Case menu.

To set the section type for a member or group of members

- **Select the required members in the Frame window**
- **Choose Section Type… from the Frame menu**



United States sections library shown

- **Choose the section from the list**
- **Click OK**

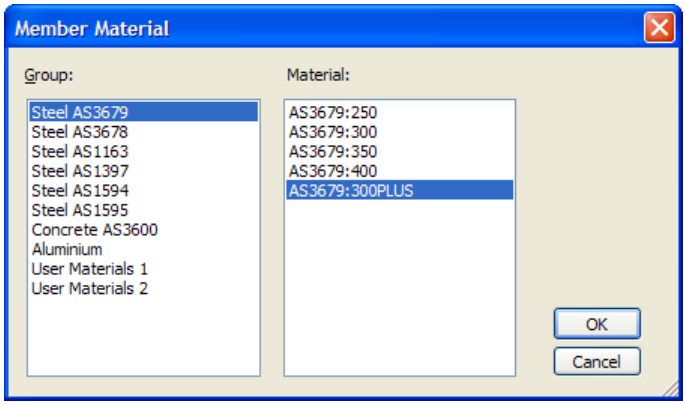
Setting Steel Grade

To determine the allowable stresses or design capacities for a member, it is necessary to know the grade of steel to be used for the section. This grade determines the yield strength (Fy) and ultimate tensile strength (Fu) of the material of the section. The strength of the steel may be specified by assigning a material, choosing a standard steel grade supported by the current design code or by specifying the values of the Fy and Fu directly.

The Japanese AIJ code does not require the ultimate tensile strength (Fu) but instead requires the user to specify the yield strength (Fy) for steel thicknesses of less than and greater than 40mm.

To set the material for a member or group of members

- **Select the required members in the Frame window**
- **Choose Member Material… from the Frame menu**



United States sections library shown

- **Choose the material from the list**
- **Click OK**

Note that if the elastic properties of the new material differ from the original material then you will need to re-analyse the structure using the Analyse command from the Case menu.

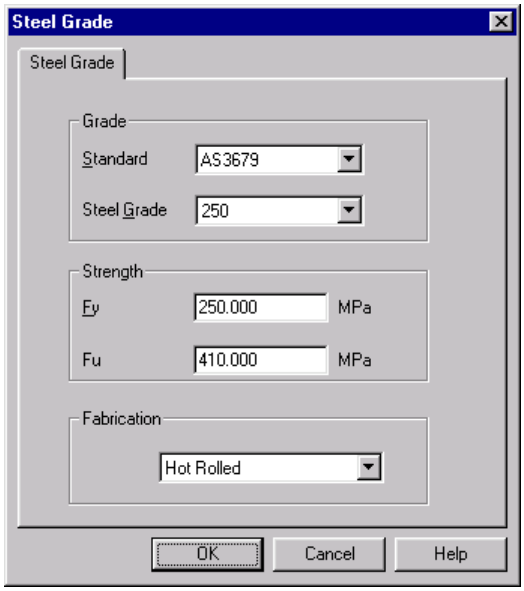
Important: Using Materials

When a material is assigned to a member Multiframe Steel Codes will try to match the material to one of the standard steel grades supported by the current design code. In this way, the design checks performed by Multiframe Steel Codes are able to take advantage of clauses that refer to specific steel grades (e.g. yield strengths that vary with thickness). All design properties, including ultimate and yield strengths, will be obtained from values specified within the design code.

If a material is not matched to a standard steel grade then the values of the yield and ultimate strength will be obtained from the material instead of from the design code. Furthermore, clauses that refer to specific design clauses will not be enacted. In cases where a material does not match to a standard steel grade it is recommended that the steel grade be assigned directly as described below.

Alternatively, to set the Steel Grade directly and override the properties of the material

- **Select the required members in the Frame window**
- **Choose Steel Grade from the Design menu**



AS4100 shown

Either

- **Choose a standard and/or steel grade from the pop-up menu or…**
- **Type in values for Fy and Fu**
  (or Fy<40mm and Fy>40mm when using AIJ)
- **Choose the fabrication type for the section**
- **Click OK**

If you choose a standard and/or a grade of steel, the Fy and Fu values will be automatically entered for you.

If no material has been assigned to a member then the initial value for the steel grade for all members is:

Code	**Grade**	**Fy**	**Fu**
ASD & LRFD	A36	36ksi	58ksi
AS4100	AS3679 grade 250	250MPa	410MPa
NZS3404	AS3679 grade 250	250MPa	410MPa
BS5950	S235	235MPa	340MPa
User (US)	-	36ksi	58ksi
User (Australia)	-	250MPa	410MPa
User (New Zealand)	-	250MPa	410MPa

Code	Grade	Fy<40mm	Fy>40mm
AIJ	SS400	2.4t/cm^2	2.2t/cm^2
User (Japan)	-	2.4t/cm^2	2.2t/cm^2

Setting Design Constraints

Steel Design uses the concept of Design Constraints to describe any design requirements that are not dependent upon the design actions and can be tested independently of the load cases. Design Constraints include constraints that may be imposed by the designer upon the dimensions of a member as well as any constraints that may be imposed by various design checks. (i.e. a slenderness check that may be required as part of a bending design).

Design Constraints are applied when Designing and Checking a member. The calculations associated with Design Constraints are output to the design report. These calculations are performed at the start of the design before considering the design checks for each load case. When using Brief Reporting, the calculations for failed design constraints are output to the report. With detailed or full reporting, the calculations for all Design Constraints are shown in the report.

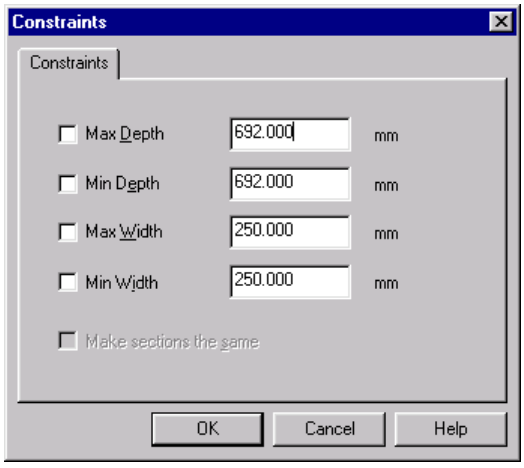
The status of Design Constraints which were tested when Designing or Checking a member are displayed in the "Constraints" column in the Design Efficiency table. If no constraints were checked for a particular member, a dash is shown is this column. Otherwise, this column displays the number of Design Constraints that were not satisfied as part of the design checks.

Setting Section Constraints

When designing a member to determine the lightest weight section that may be used, you may wish to apply some constraints to the way the sections are selected. For example, you may wish to limit the section's depth or width or you may wish to ensure that a group of members all use the same section.

To constrain the selection of a member's section

- **Select the required members in the Frame window**
- **Choose Constraints… from the Design menu**



- **Check the boxes corresponding to the sizes you wish to constrain**
- **Type in the limits for the sizes you wish to constrain**
- **If you wish to make the sections the same, check the "Make sections the same" check box**

- **Click OK**

The initial value of constraints is for no limits on the sizes of sections and all members are free to be designed using a different section.

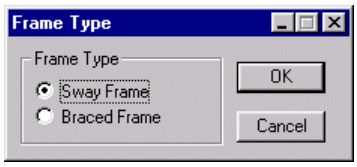
Variable Name	Description	Default
Max Depth	The maximum depth of section which may be chosen when using the Design command	Depth of the initial section
Min Depth	The minimum depth of section which may be chosen when using the Design command	Depth of the initial section
Max Width	The maximum width of section which may be chosen when using the Design command	Width of the initial section
Min Width	The minimum width of section which may be chosen when using the Design command	Width of the initial section

Setting Frame Type

Some design calculations depend on whether the frame is free to deflect laterally (sway) or is restrained by internal or external bracing to prevent side-sway (braced). A sway frame develops all of its horizontal stiffness due to the flexural actions of the columns in the structure. In contrast, the bracing in a braced frame absorbs the horizontal forces and horizontal deflections of the columns are reduced to a minimum.

To set the type of frame

- **Choose Frame Type from the Design menu**



- **Click on type of the frame**
- **Click OK**

The initial setting for the frame type is a sway frame.

Setting Allowable Stresses

Some steel design codes permit you to increase the allowable stresses by a set amount (usually 33 or 50%) for load cases that only involve temporary loading. Multiframe Steel Codes allows you to utilize this option by using the Allowable Stresses option from the Design menu. This allows you to enter a factor for the allowable stress increase for each load case.

The initial value of the allowable stress increase factor is 1.0 for all load cases. If, for example, you wanted the stresses for a load case to be allowed to increase by 33%, you would enter a value of 1.33.

Setting Acceptance Ratio

Some of the design codes within Multiframe Steel Codes allow the user to modify the value of the efficiency below which the design checks on a member have deemed to of passed. This value is known as the Acceptance Ratio. Any design check on the member for which the efficiency exceed this value will be marked as a failed check.

The Acceptance ratio for a particular member is set via the Options command in the Design menu. The initial value of the Acceptance Ratio for all members is 100%.

Setting Capacity Factors

In limit state design the design capacity is obtained by multiplying the nominal capacity by the capacity factor. The capacity factor will vary depending upon the specific design check being considered. The design codes generally specify maximum values for the capacity factors. In some circumstances the user may wish to specify other values for the capacity. Multiframe Steel Codes allows you to do this by using the Capacity Factors option from the Design menu. A dialog is displayed which allows the user to change the capacity factors for each of the design checks for a strength limit state.

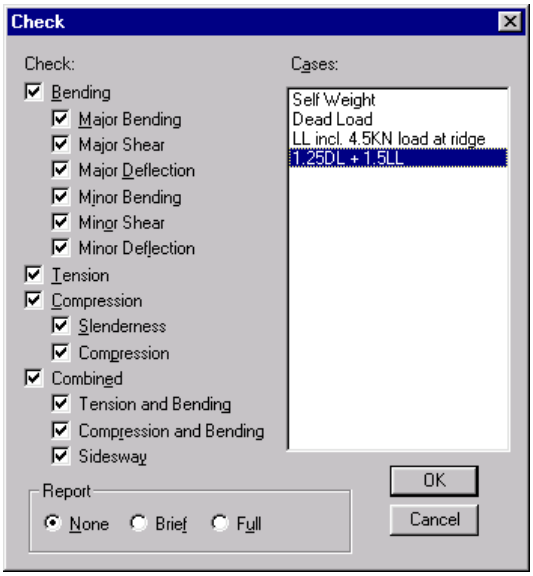
The initial values of the capacity factors are the values specified by the design codes. In most likely that the capacity factors will never be modified by a user.

Checking a Frame

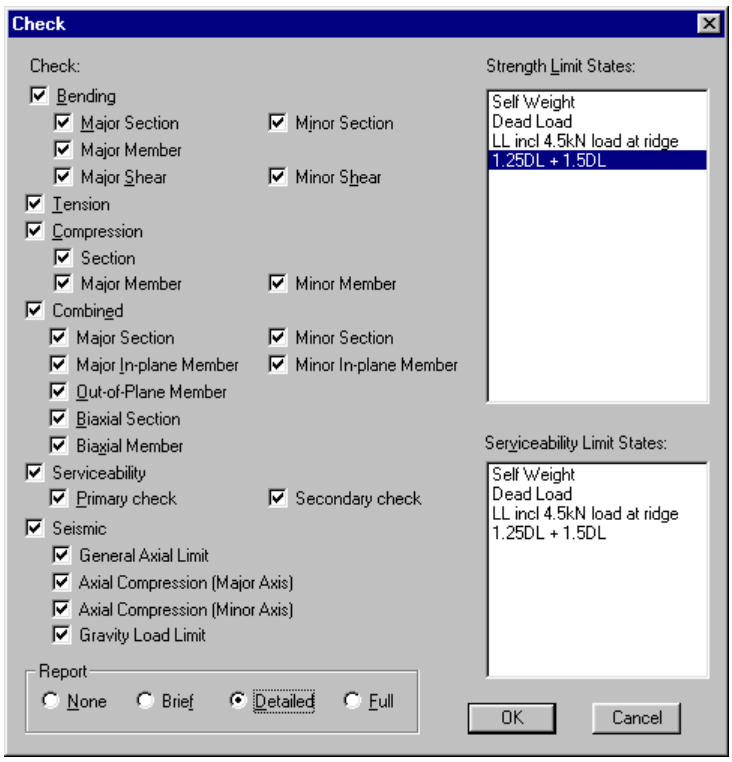
Once you have set up the structure and its design properties, you can check it for compliance with the code rules.

To check a member or group of members

- **Select the required members in the Frame window**
- **Choose Check… from the Design menu**



ASD, AIJ



AS4100, NZS3404

- **Check the boxes of the design rules to be checked**
- **Shift or Ctrl Click on the load case names in the list to include or remove them from the check**
- **If you want a summary report in the Report window, check the Brief, Detailed or Full report radio buttons**
- **Click OK**

Multiframe Steel Codes will work through the selected members checking the stresses for the load cases you have chosen for compliance with the design rules you specified. The result of the check for the current load case will be displayed in the Design Efficiency table in the Result window. Each column in this table shows the member's strength as a percentage of the allowable strength according to the code. For example, an efficiency of 95% means that the member is being stressed to 95% of its allowable value. An efficiency greater than 100% indicates that the member is being stressed to a higher level than that permitted by the code. The Overall column shows the highest value of all of the design checks for the member for the current load case. The subsequent columns show the result for the individual checks, which have been carried out.

You can display the results for different load cases by choosing the appropriate item from the Case menu.

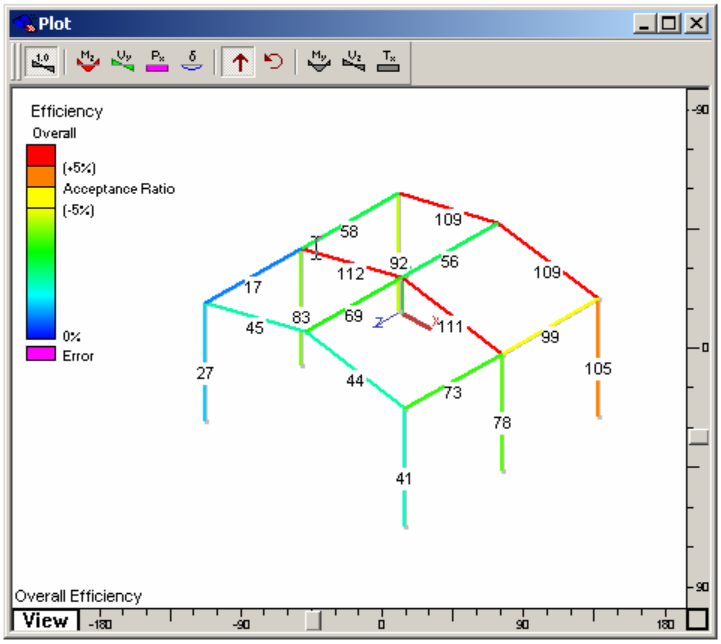
The check will be much slower if you choose to have a summary report generated, however the report will contain detailed information about all of the design checks carried out. You will probably find it best to do an overall check on the areas of interest without the report on and then check a few key members using the full report option.

Displaying Efficiency

As well as displaying the table of member efficiency in the Result window, you can view these values graphically in the Plot window.

To view the member efficiency

- **Choose the required item from the Efficiency sub-menu under the Display menu**



The members will be drawn in the Plot window with a colour code indicating the efficiencies of the members. The scale shown in the legend may be used to determine the relative values of the colours. Members, which exceed the allowable capacity, will have an efficiency greater than acceptance ratio for the member (typically 100%) and will be drawn in orange or red.

If you turn on the display of Plot Values in the Symbols dialog under the Display menu, the values of the efficiencies will be displayed on the members.

Values and colours will only be drawn for members, which have been checked. You can also use the clipping and masking commands to restrict which members have their efficiency values displayed.

Governing Load Cases

The governing load case associated with the overall design of a member is recorded when designing or checking a member. The governing load case associated with each member is displayed in the Efficiency table in the Result Window.

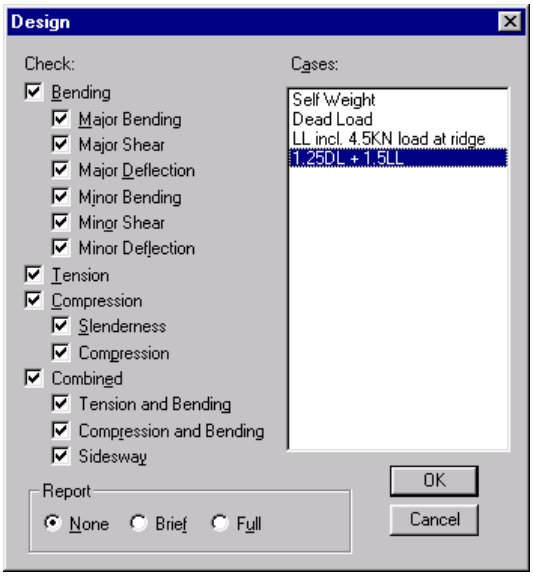
The load cases governing the design of each of the individual design checks are also recorded when designing or checking a member. The governing load case for a specific design check can be displayed in two ways: as a cell tool tip in the Efficiency table or as a member tool tip in the Plot Window when plotting the efficiency of the particular design check.

Designing a Frame

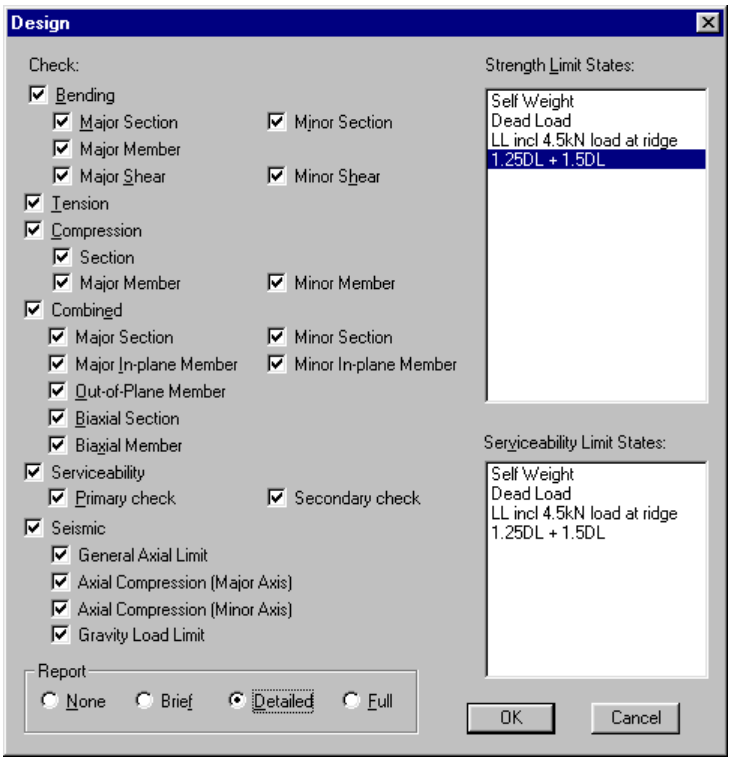
As well as helping to check a frame's compliance with the design rules, Multiframe Steel Codes can also help you to select the lightest weight section that satisfies the design rules.

To design a member or group of members

- **Select the required members in the Frame window**
- **Choose Design… from the Design menu**



ASD, AIJ



AS4100, NZS3404

- **Check the boxes of the design rules to be used when designing**
- **Shift-Click on the load case names in the list to include or remove them from the check**
- **If you want a summary report in the Report window, check the Brief or Full report radio buttons**
- **Click OK**

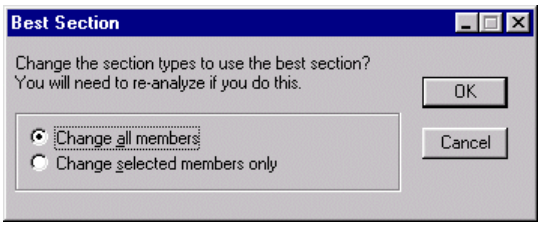
Multiframe Steel Codes will design each of the selected members; searching through the group of sections the member's original section comes from, to find the lightest section in this group that meets the design rule requirements. Once the design has finished, you can view the optimum section in the Best Section column in the Member Efficiency table in the Result window. If you want to automatically assign all of the optimum sections to their respective members, you can use the Use Best Sections command from the Design menu to do this. Because changing the sections will change the results of the analysis, you will have to re-analyse the structure after doing this. You may find it useful to wait until you have designed all of the members you wish to optimise before using the Use Best Sections command.

Optimum Sections

Once you have computed an optimum weight section for a member using the Design command, the best section will be displayed in the Design Efficiency table in the Result window. You can refer to this table to compare the optimal section with the original section. If you decide that you want to permanently replace the original section with the best section you should use the Use Best Sections command from the Design menu. If you have selected members in the front window you can choose to only update the selected members or you can update the entire frame. In any case, only members, which have been designed, will be updated.

To change sections to the optimum sections designed

- **Choose Use Best Sections from the Design menu**


Best Section
Change the section types to use the best section?
You will need to re-analyze if you do this.
OK
Cancel
Change all members
Change selected members only

- **Click the radio button to change just the selected members or the entire frame**
- **Click OK**

The sections of the member’s chosen will be changed to the optimal sections. After using this command you will have to re-analyse the frame to determine the effect of your change on the structure.

The user can override the design and specify the optimal section for a member using the command from the Design menu in which case the select section dialog will be displayed. As this command does not invalidate the results of analyses it can be used to temporarily store the next section shape to be allocated to a member. In this way other members in the frame can be investigated before having to reanalyses the structure.

Tips On Optimisation

When you use the Design command, Multiframe Steel Codes will try to find the lightest weight section in a member's group, which will satisfy the design requirements. If there are a large number of sections in the group, this may take some time. If you use the options to constrain the width or depth of the optimum section, Multiframe Steel Codes will automatically skip the check for any sections, which don't satisfy these criteria. This means you can speed up the optimisation greatly by specifying constraints for the size of the section. For example, if you are selecting an optimum section from the W sections in the United States Section Library which contains a large number of sections, specifying an upper and lower bound for the depth will let Multiframe Steel Codes automatically skip most of the sections and quickly find one of the right size.

Checking for sway when using the Design command is not recommended. It is unlikely that Multiframe Steel Codes will find an optimum size member because the amount of sway is likely due to the stiffness of other members (probably the columns in another part of the frame) rather than the member under consideration. These other members will not be changed while the current member is being checked.

Finding Design Values

The Find command from the Edit menu can be used to automatically search through the structure to find members that have design values exceeding a specified value for the current load case. You can search for actions, deflections, stresses or efficiencies.

To search for a category of members

- **Choose Find from the Edit menu**

- **Click on the pop-up menu to choose the category to search for**
- **Click on the radio buttons to set the criteria for the search**
- **Click OK**

After searching through the frame, Multiframe Steel Codes will select all of the members, which meet the specified criteria.

Printing

You can print the contents of any of the windows including the Report window.

Printing the Report Window

To print the contents of the Report window

- **Ensure the Report window is in front**
- **Choose Print Window from the File menu**

As with the other windows in Multiframe, the user may review the output in the Print Preview before sending the output to the printer.

Saving your Work

You can save your design work at any time and then open the frame later to continue where you left off.

To save the frame and its design information to disk

- **Choose Save from the File menu**

The frame will be saved to disk complete with the design information you added to it.

Saving the report

You can also save the report to disk and recall it at a later date.

To save the report to disk

- **Ensure the Report window is in front**
- **Choose Save from the File menu**

The report will be saved to disk. Use the Open command to read the report in again. If you need to transfer the data in the report to another program like Microsoft Word, use the Select All and Copy and Paste command to paste the data into the other program. Multiframe Steel Codes places the report data on the clipboard in the RTF (Rich Text) format.

Chapter 3
ASD and AIJ

This chapter describes the implementation of the ASD and AIJ steel design codes within Multiframe Steel Codes. It provides a step-by-step description of how to modify the design properties used by each code.

- Design Checks
- Bending
- Tension
- Compression
- Combined Actions
- Default Design Properties
- Code Clauses Checked

Design Checks - ASD and AIJ

The design checks performed using the ASD and AIJ codes are grouped into the four categories; Bending, Tension, Compression, and Combined.

Bending - ASD and AIJ

There are six design checks grouped under the Bending category. These checks verify a member's capacity to resist bending moments and shear forces about the major and minor axes. Design checks for the deflection of the member are also included in this group.

When performing a bending check, you need to specify a number of properties relating to the unbraced length and the spacing of stiffeners on the member. When using the ASD code, the user may also specify a bending coefficient.

Design Constraints (AIJ)

When checking or designing a member for bending, compression or combined bending and compression, a design constraint is automatically imposed by Multiframe Steel Codes. This constraint verifies that the member satisfies the requirements of AIJ for the Width to Thickness Ratio (b/t) of Plate Elements.

Unbraced Length - ASD and AIJ

To determine the critical buckling condition of a member, it is necessary to know the spacing of any bracing (if any) along the member. Purlins, girts or other structural elements that are not modelled in Multiframe could provide this bracing. Some bracing may only restrain lateral deflection in one direction. It is therefore necessary to enter unbraced lengths for both axes of the section, Lbx corresponding to the spacing of restraints preventing buckling about the x-x axis and Lby corresponding to the spacing of restraints preventing buckling about the y-y axis.

The initial values of Lbx and Lby are the length of the member.

Bending Coefficient (ASD)

The ASD code requires a bending coefficient Cb that is either calculated by the program according to the rules in the code, or may be specified by the user. If you leave Cb unchanged, Multiframe Steel Codes will select a value for you, which will be displayed in Italics in the Design Details table in the Data window. This value is most commonly 1.0. If you type in a value, Multiframe Steel Codes will always use this value and display it in non-italic (i.e. standard) text in the Design Details table.

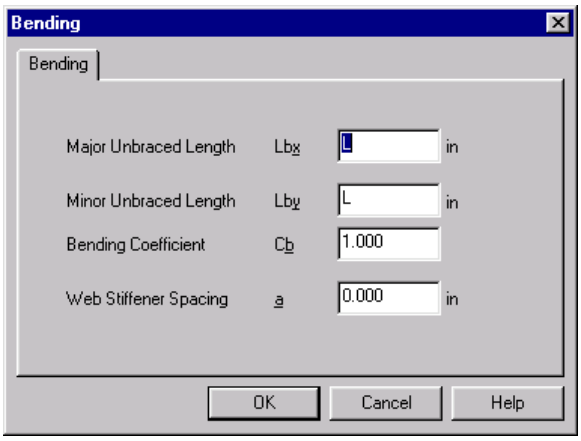
Web Stiffener Spacing - ASD and AIJ

When checking or designing a member for bending, you may need to specify the spacing of any stiffeners along the web of the member. This affects the member's susceptibility to buckling due to bending. If there are no transverse stiffeners, you should leave the stiffener spacing set to zero.

Bending Dialog - ASD and AIJ

To set the properties for bending

- **Select the required members in the Frame window**
- **Choose Bending from the Design menu**



- **Type in values for Lbx and Lby**
- **If necessary enter a value for the bending coefficient Cb**
- **Type in the stiffener spacing (s)**

Tension - ASD and AIJ

The capacity of a member to resist tensile forces is implemented as a single design check. A number of modification factors may be entered to change the section properties used for checking tension. This includes the area of holes in the cross section of the member and an area reduction coefficient used to compute the effective area of the section.

Bolt Holes - ASD and AIJ

When checking or designing a member for tension, you need to specify any reduction in area due to boltholes or other reductions. If the members contain significant areas of boltholes, which need to be taken into account when determining the cross-sectional area of the section, you will need to enter the amount of cross-sectional area to be deducted to allow for these holes. The initial value for the area of boltholes is zero.

The net area of the section is the gross area minus the combined area of boltholes in the flange and web.

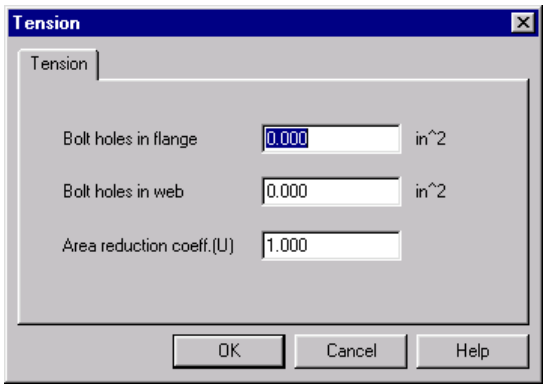
Area Reduction - ASD and AIJ

The net area is multiplied by the area reduction coefficient, U, to give the effective net area of the section. The default value of U is 1.0, i.e. no reduction in area.

Tension Dialog - ASD and AIJ

To enter the properties for tension

- **Select the required members in the Frame window**
- **Choose Tension… from the Design menu**



- **Type in the area of holes in the web and flanges**
- **Type in a value for the area reduction coefficient (U) if required**

Compression - ASD and AIJ

Multiframe Steel Codes splits the compressive design of a member into two design checks. You may choose to check the slenderness of a member and/or its compressive stress.

When checking or designing members for compression, it is necessary to specify the effective length and unbraced length of the member.

To determine the critical buckling load for a member, it is necessary to enter an effective length to indicate the type of restraint on the ends of the member. The effective length is given by an effective length factor multiplied by the length of the member. The effective length may be different for buckling in the major and minor axis directions. The effective lengths are given by

Lx=Kx*L and Ly=Ky*L

Where L is the length of the member and Kx and Ky are the two effective length factors for the major and minor axes respectively.

The initial values of Kx and Ky are 1.0.

The slenderness is measured as:

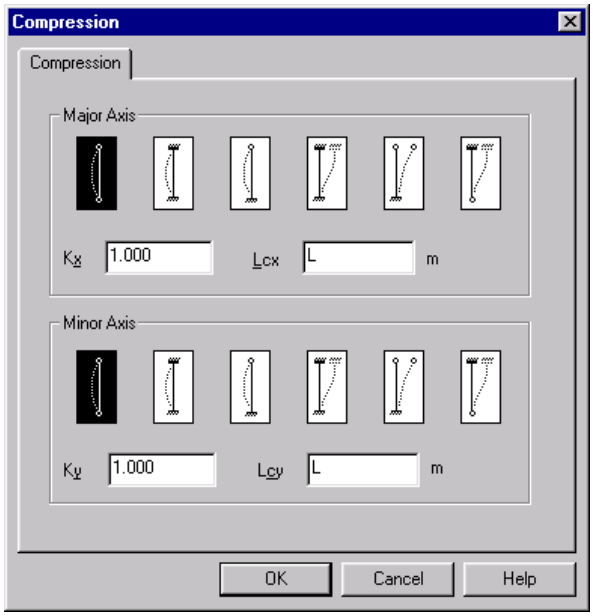
$$\text{Slenderness} = \text{Maximum of} \left\{ \begin{matrix} Kx*L/rx \\ Ky*L/ry \end{matrix} \right.$$

See also: Unbraced Length

Compression Dialog - ASD and AIJ

To set the properties for compression

- **Select the required members in the Frame window**
- **Choose Compression… from the Design menu**



Either

- **Click on the icons for the end conditions in each direction**

Or

- **Type in values for Kx and Ky**
- **Type in values for Lcx and Lcy**
- **Click OK**

If you choose a standard end condition, the recommended Kx and Ky values will be automatically entered for you.

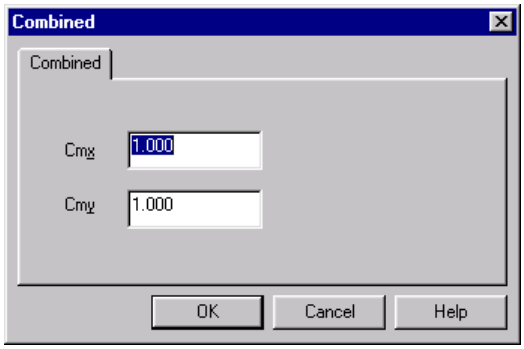
Combined Actions - ASD and AIJ

When a member is subject to bi-axial bending or a combination of axial tension or compression and bending, it is likely to be necessary to carry out a combined check on the member's performance as a beam-column. This combined check usually takes the form of a comparison of the sum of the ratios of the actual stress to the allowable stress for each of the considered actions. As columns are frequently subject to these types of actions, there is also an option to check the side sway of a beam-column. The side sway check usually takes the form of a comparison of the horizontal deflection at the top of the member with a proportion of its height above ground level.

When checking or designing members for combined bending and compression actions under the ASD code, you may wish to enter coefficients as prescribed by the code. If you leave the Cm unchanged, Multiframe Steel Codes will select a value for you, which will be displayed in italics in the Design Details table in the Data window. This value is most commonly 1.0.

To set the coefficients for combined checks

- **Choose Combined… from the Design menu**



- **Enter the values for Cmx and Cmy**
- **Click OK**

Default Design Properties - ASD and AIJ

There are a number of design variables, which are used when doing checking to the code. A summary of all of the design variables is as follows;

Variable	Description	Default
Fy	Yield strength of the section's steel	36ksi
Fu	Ultimate Tensile Strength of the section's steel	58ksi
Kx	Effective length factor for buckling about the section's strong axis	1.0
Ky	Effective length factor for buckling about the section's weak axis	1.0
Lbx	Unbraced length for bracing preventing buckling about the section's strong axis	Member’s length
Lby	Unbraced length for bracing preventing buckling about the section's weak axis	Member’s length
a	Spacing of web stiffeners. This is the spacing of any stiffeners along the web of a beam	0.0 (i.e. no stiffeners)

Flange Hole Area	Area of any bolt holes in the flanges of the section. This area will be deducted from the cross sectional area when computing tensile stress	0.0
Web Hole Area	Area of any bolt holes in the web of the section. This area will be deducted from the cross sectional area when computing tensile stress	0.0
U	Area Reduction factor. This factor is applied to the sectional area (after bolt holes have been deducted) when calculated tensile stress. You can use it to reduce the effective area by a defined amount. It must have a value between 0 and 1.0	1.0
Cb	Moment modification factor used to determine allowable compressive stresses in bending. (See ASD code for details)	1.0
Cmx	Moment reduction coefficient for bending about the section's strong axis (see ASD code)	1.0
Cmy	Moment reduction coefficient for bending about the section's weak axis (see ASD code)	1.0
Fabrication	The method by which the section was manufactured. This describes the residual stresses in the section.	Rolled

Code Clauses Checked - ASD and AIJ

When carrying out code checks, Multiframe Steel Codes uses the following clauses of the applicable codes to check your structure. No other checks are performed unless they are specifically listed below.

ASD Clauses Checked

"Specification for Structural Steel Buildings, Allowable Stress Design and Plastic Design", American Institute of Steel Construction, June 1, 1989 (contained in Manual of Steel Construction, Allowable Stress Design, 1989, 9th Edition).

Clauses used are A5.1, A5.2, B1, B3, B5, B7, C2, D1, E1, E2, F1, F2, F3, F4, G1, G2, G3, H1, H2

The design checking procedure is as follows;

The section is classified and tensile area and limiting slenderness ratios are determined according to section B.

For major and minor bending checks, the bending stress is checked to be less than the allowable Fb as found in sections F1, F2 and F3.

For major and minor shear, the shear stress is checked to be less than the allowable Fs found from section F4. The shear stress is computed using a shear area as shown above.

For major and minor deflection due to bending, the maximum deflection is checked to be less than L/300. No specific check is made for cantilevered members.

For tension checks, the tensile stress is checked to be less than the allowable Ft on both the gross and net areas as computed in section D1.

For slenderness checks, the slenderness ratio is computed as the maximum of KxL/rx and KyL/ry. This is checked to be less than the allowable slenderness ratio of 200 for compressive members or 300 for tensile members in accordance with clause E1.

For compression checks, the compressive stress is checked to be less than the allowable Fa as computed in section E2.

For combined compression and bending checks, the stresses are checked to be low enough to satisfy equations H1-1 to H1-3.

For combined tension and bending checks, the stresses are checked to be low enough to satisfy equation H2-1.

For sway checks, the horizontal deflection of the highest part of the member is checked to be less than Y/300 where Y is the height of the highest part of the member above the plane y=0.

Checks are not carried out on hybrid members, composite members or tapered members.

AIJ Clauses Checked

"Design Standard for Steel Structures", Architectural Institute of Japan, March 1979.

Clauses used are 5.1, 5.6, 6.1, 6.2, 8.1, 10.1, 11.1, 11.2, 11.3

The design checking procedure is as follows;

Allowable stresses are determined from table 5.1 and according to equations 5.1, 5.2, 5.3, 5.4, 5.5, 5.6, 5.7 and 5.8 as appropriate.

For major and minor bending checks, the width-thickness ratio of the section's elements are checked in accordance with equations 8.1, 8.2, 8.3, 8.5 and 8.6 as appropriate. The bending stress is checked to be less than the allowable fb as found in section 5.1.4.

For major and minor shear, the shear stress is checked to be less than the allowable fs found from equation 5.2. The shear stress is computed using a shear area as shown above.

For major and minor deflection due to bending, the maximum deflection is checked to be less than L/300 in accordance with clause 10.1. No specific check is made for cantilevered members.

For tension checks, the tensile stress is checked to be less than the allowable ft as computed using equation 5.1.

For slenderness checks, the slenderness ratio is computed as the maximum of KxL/rx and KyL/ry. This is checked to be less than the allowable slenderness ratio of 200 for vertical members or 250 for non-vertical members in accordance with clause 11.2 (A vertical member is assumed to be one which is within 100mm of vertical).

For compression checks, the compressive stress is checked to be less than the allowable fc as computed in equation 5.3 or 5.4.

For combined compression and bending checks, the stresses are checked to be low enough to satisfy equations 6.1 and 6.2.

For combined tension and bending checks, the stresses are checked to be low enough to satisfy equations 6.3 and 6.4. The area of bolt holes as specified in the Bolt Holes dialog is deducted from the gross section area to calculate the net section area.

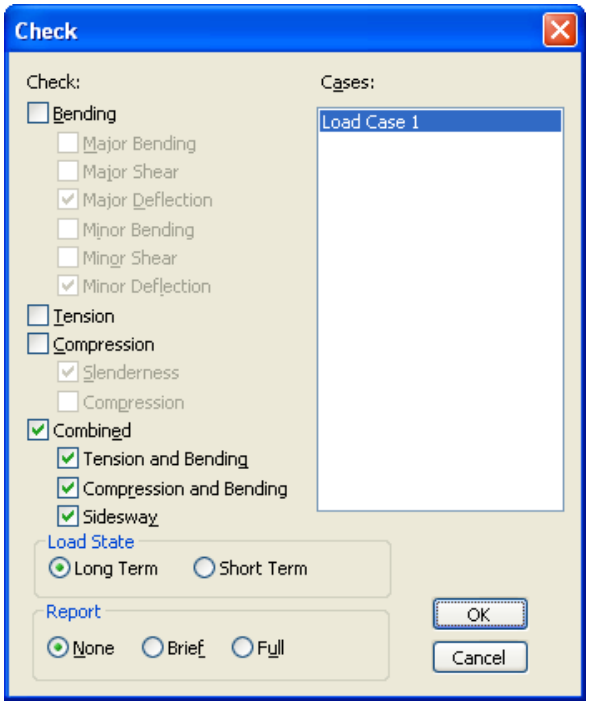
For sway checks, the horizontal deflection of the highest part of the member is checked to be less than H/300 where H is the height of the highest part of the member.

Short Term Loads for AIJ

As defined in the AIJ code, if the loads are short term the allowable strength if increased by 50%. To define the loads as short term click the Short Term radio button in the Load State section of the AIJ Design Check dialog.

To define the loads as short term

- **Ensure the AIJ code is chosen (Design -> Code -> AIJ)**
- **Select one or member**
- **Choose Check… from the Design menu**
- **Select Short Term from the Load State Group in the dialog shown below**
- **Click OK to run the design check**



Chapter 4
AS4100 and NZS3404

This chapter describes the implementation of the Australian AS4100 and New Zealand NZS3404 steel design codes within Multiframe Steel Codes. It provides a step-by-step description of how to modify the design properties used by each code.

- Notation
- Design Checks
- Bending
- Tension
- Compression
- Combined Actions
- Serviceability
- Seismic (NZS3404)
- Default Design Properties
- Code Clauses Checked

Notation - AS4100 and NZS3404

The notation used in Multiframe Steel Codes generally follows that used in AS4100 and NZS3404. There are some minor differences that are noted below. In addition, some extra notation has been introduced to help clarify the different design quantities.

kte	Correction factor for distribution of forces in a tension member (equivalent to kt in AS4100).
Ncx1	nominal member capacity in axial compression for buckling about the major principle axis computed using a maximum effective length factor (ke) of 1.0.
Ncy1	Nominal member capacity in axial compression for buckling about the minor principle axis computed using a maximum effective length factor (ke) of 1.0.

Design Checks - AS4100 and NZS3404

The types of checks are grouped into the categories; Bending, Tension, Compression, Combined, Serviceability and Seismic (NZS3404 only). The user may specify which of these checks are performed when a member is designed or checked using Multiframe Steel Codes.

Bending - AS4100 and NZS3404

The design of a member for bending consists of five design checks. These check the section capacity of the member about the major and minor axes, the shear capacity about both axes and the member, or buckling, capacity about the major axis.

When performing a bending check it is necessary to specify how lateral buckling of the member is resisted. Restraint could be provided by other members, purlins, girts or by other structural elements that are not modelled in Multiframe such as concrete slabs. Multiframe Steel Codes provides three methods of specifying how a member is restrained against lateral buckling. The user may specify

That the member is fully restrained against lateral buckling in which case no lateral buckling checks will be performed.
The location and type of lateral restraints applied to the member in which case Multiframe Steel Codes will appropriately divide the member into a number of spans and consider the capacity of each of these spans in determining the capacity of the member.
The laterally unbraced length (l_e) and moment modification factor (α_m).

You may need to specify a number of properties relating to the location and type of lateral restraints and the stiffener spacing along the member

Lateral Restraints - AS4100 and NZS3404

To determine the moment member capacity of a member, it is necessary to know the spacing of any lateral restraints (if any) along the member. The restraints could be provided by purlins, girts or other structural elements, which are not modelled in Multiframe. Multiframe Steel Codes uses this information to determine the length of segments used in the design calculations. The lateral restraints acting at a particular section on a member are dependent upon which flange is the critical flange. For a member/segment restrained at both ends the critical flange is the flange under compression. For a cantilever or a segment with an unrestrained end, the critical flange is the tension flange. For each restraint on the member, the user must specify the type of restraint. As this depends upon which flange is the critical flange, the user must specify the type of lateral restraint that would be present at a section if

i) The top flange were the critical flange, and
ii) The bottom flange was the critical flange.

Lateral restraints must always be specified at the ends of the beam and so the minimum number of lateral restraints is two. If no restraint exists at the end of a member then it should be specified as unrestrained. The initial lateral restraints applied to the member are full restraints at each end for either of the flanges being the critical flange.

The different restraints acting on the member can be specified as;

Restraint Type	Abbreviation
Fully restrained	F
Partially restrained	P
Laterally Restrained	L
Unrestrained	U
Continuous restraint	C

Fully or partially restrained sections may also be specified as lateral rotational restraints using;

Restraint Type	Abbreviation
Fully restrained and Rotationally restrained	FR
Partial restrained and Rotationally restrained	PR

The initial position of the loads is at the shear centre. If there are no transverse stiffeners, leave the stiffener spacing set to zero.

The location and type of lateral restraints can be displayed in the Frame and Plot windows. The display of lateral restraints can be turned on or off via the Symbols Dialog which now contains options for displaying lateral restraints and labelling these restraints.

The restraints are draw as a short line in the plane of the major axis of the member. These lines extend each side of the member for a distance that is roughly the scale of a purlin or girt. Lateral restraints are also displayed in the rendered view of the frame in which they are draw to extend from each flange by approximately the size of a purlin. The restraints may be labelled using a one or two letters to indicate the type of restraint (e.g. F - fixed, P - partial).

Note that lateral restraints at the end of a member are draw slightly offset from the node so that restraints at the ends of connected members may be more readily distinguished.

Unbraced Length (l_e) and Bending Coefficient (α_m) - AS4100 and NZS3404

Instead of specifying the position of lateral restraints it may be preferable to directly set the laterally unbraced length of the member. When doing this, it is also necessary to specify the bending coefficient (α_m) as this can no longer be automatically determined by Multiframe Steel Codes. The design codes permit a conservative value of α_m=1.0 to be adopted which is the default value used by Multiframe Steel Codes.

Web Stiffener Spacing - AS4100 and NZS3404

When checking or designing a member for bending, you may need to specify the spacing of any stiffeners along the web of the member. This affects the member's susceptibility to buckling due to bending. If there are no transverse stiffeners, you should leave the stiffener spacing set to zero.

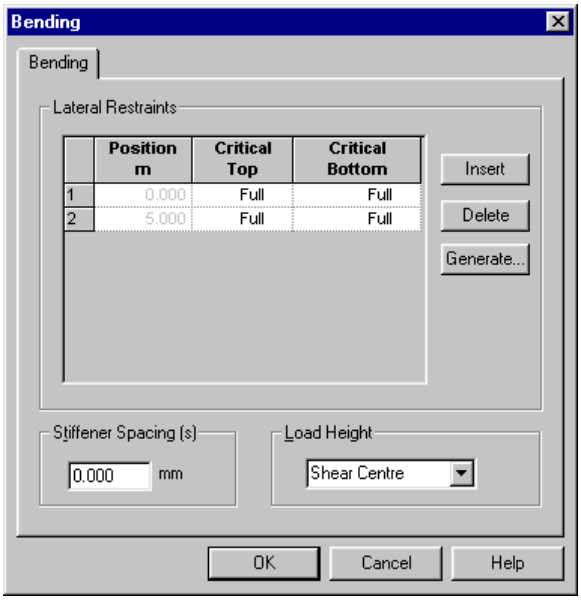
Load Height - AS4100 and NZS3404

When checking or designing a member for bending, you may need to specify the load height position. This is used in determining the effective lengths of segments or sub-segments along the member.

Bending Dialog - AS4100 and NZS3404

To set the properties for bending

- **Select the required members in the Frame window**
- **Choose Bending from the Design menu**



If the member is fully braced against lateral torsion buckling

- **Select the "Member is fully laterally restrained" option**

or if the location of lateral bracing along the member is to be specified

- **Select the "Position of Lateral Restraints" option**

To add new restraint to the member

- **Position the cursor with the table and click the Insert button to add a lateral restraint to the member.**
- **Select the position of each restraint**
- **Select the type of each lateral restraint from the combo provided in each cell.**

or

- **Click the Generate button to automatically generate a number of restraints.**

To delete a restraint from the member

- **Position the cursor within the table on the lateral restraint to be deleted and click the Delete button.**

or if the unbraced length of the member if the be specified directly

- **Select the "Unbraced Length" option**
- **Enter the unbraced length (l_e)**
- **Enter the moment modification factor coefficient (α_m) to be used in the design of this length of the member.**

And then

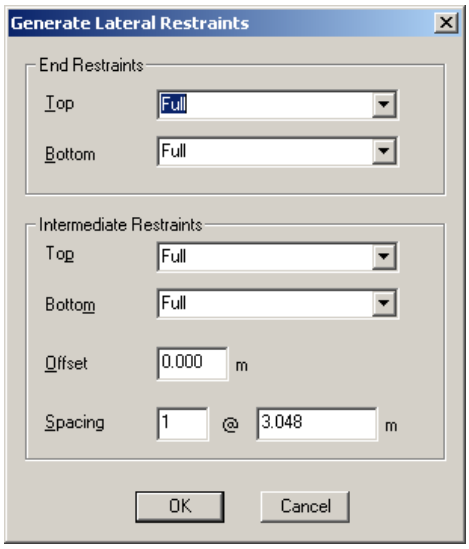
- **Choose the position of the load from popup menu**
- **If there are transverse stiffeners on the web, type in values for the stiffener spacing (s)**

- **Click OK**

Generate Lateral Restraints Dialog - AS4100 and NZS3404

When the user selects to generate the lateral restraints from the Bending dialog, the Generate Lateral Restraints dialog is displayed. This dialog enables the user to generate lateral restraints are a specified spacing along the member.

- **From the Bending dialog, click the Generate… button**



- **Select the type of restraints to be used at the ends of the member**
- **Select the type of restraints to be used at intermediate points within the member**
- **Enter the offset length at which the first intermediate restraint will be positioned. Leave this field as zero if no offset is same as the spacing**
- **Enter the number and size of spacings for the intermediate restraints.**
- **Click OK**

All lateral restraint applied to the member will now be regenerated and will replace all existing restraints.

Tension - AS4100 and NZS3404

The capacity of a member to resist tensile forces is implemented as a single design check. A number of modification factors may be entered to change the section properties used for checking tension. This includes the area of holes in the flange or web of the member and a correction factor to account for the distribution of forces at the ends of a member.

Bolt Holes - AS4100 and NZS3404

When checking or designing a member for tension, you need to specify any reduction in area due to boltholes or other openings within the section. If the members contain significant areas of boltholes, which need to be taken into account when determining the cross-sectional area of the section, you will need to enter the amount of cross-sectional area to be deducted to allow for these holes. The net area of the section is the gross area minus the combined area of boltholes in the flange and web.

The reduction in area can be specified by setting the number and diameter of holes in the web or flanges or the member. Alternative, the user may override this and directly specify the height of holes across the flanges and webs of the cross section. These heights are multiplied by the thickness of the section to determine the total reduction in area of the section. The initial value for the area of boltholes is zero.

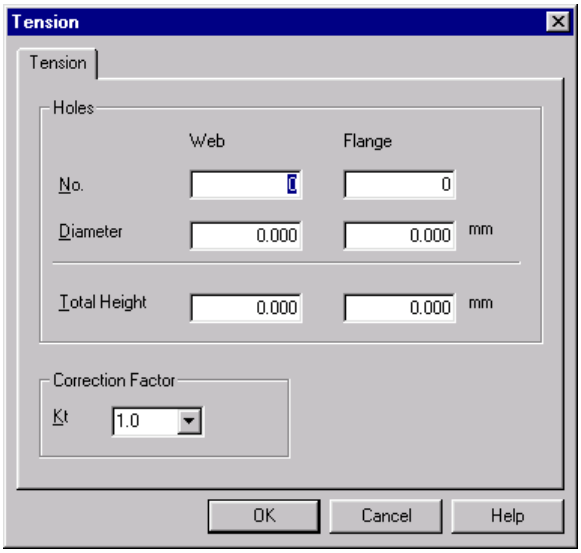
Correction Factor - AS4100 and NZS3404

When checking or designing a member for tension using AS4100 or NZS3404, you need to specify the correction factor for the distribution of forces at the ends of the member. The correction factor k_{te} has a default value of 1.0

Tension Dialog - AS4100 and NZS3404

To enter the properties for tension

- **Select the required members in the Frame window**
- **Choose Tension… from the Design menu**



- **Type in the number and diameter of holes in the webs and flanges (and the total height of holes will be computed automatically) or…**
- **Type the total height of holes in the webs and flanges directly**
- **Choose a value for the correction factor (kt) if required**
- **Click OK**

Compression - AS4100 and NZS3404

Multiframe Steel Codes splits the compressive design of a member to AS4100 and NZS3404 into three design checks. You may choose to check the section capacity and/or the member capacities about the major and minor axes.

When checking or designing members for compression, it is necessary to specify the effective length and unbraced length of the member.

To determine the critical buckling load for a member, it is necessary to enter an effective length to indicate the type of restraint on the ends of the member. The effective length is given by an effective length factor multiplied by the length of the member. The effective length may be different for buckling in the major and minor axis directions. The effective lengths are given by

Lx=Kx*L and Ly=Ky*L

Where L is the length of the member and Kx and Ky are the two effective length factors for the major and minor axes respectively.

The initial values of Kx and Ky are 1.0.

Unbraced Length - AS4100 and NZS3404

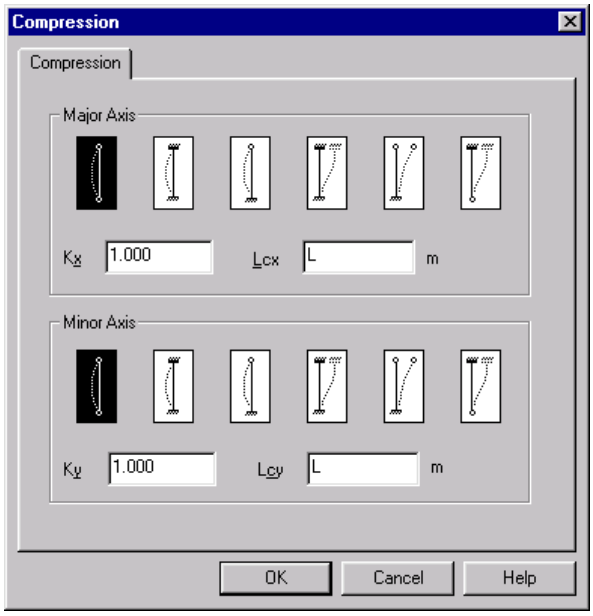
To determine the critical buckling condition of a member, it is also necessary to know the spacing of any bracing (if any) along the member. This bracing could be provided by purlins, girts or other structural elements, which are not modelled in Multiframe. Some bracing may only restrain lateral deflection in one direction, therefore it is necessary to enter unbraced lengths for both axes of the section, Lcx corresponding to the spacing of restraints preventing compression buckling about the x-x axis and Lcy corresponding to the spacing of restraints preventing compression buckling about the y-y axis.

The initial values of Lcx and Lcy are the length of the member.

Compression Dialog - AS4100 and NZS3404

To set the properties for compression

- **Select the required members in the Frame window**
- **Choose Compression… from the Design menu**



Either

- **Click on the icons for the end conditions in each direction or…**
- **Type in values for Kx and Ky**
- **Type in values for Lcx and Lcy**
- **Click OK**

If you choose a standard end condition, the recommended Kx and Ky values will be automatically entered for you.

Combined Actions - AS4100 and NZS3404

The design of a member for combined actions is divided into seven design checks. The user can select to check the section capacity and/or the member capacity about either the major and/or minor axes as well as in biaxial bending.

When using NZS3404, the combined actions checks are only performed if the member has a significant axial force as defined in the design code.

No design properties are required when checking or designing members for combined actions using AS4100 or NZS3404.

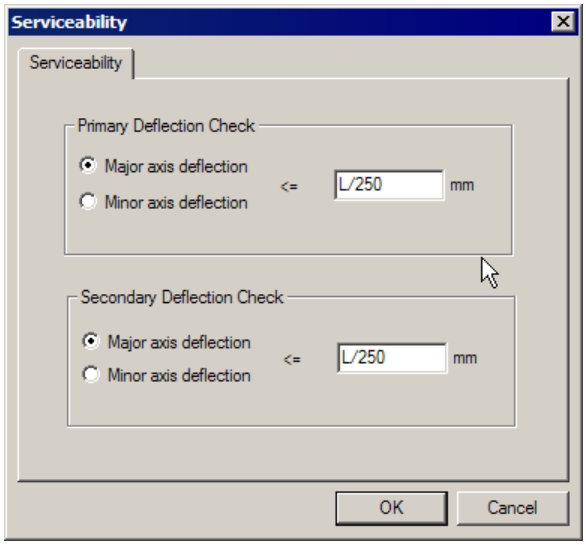
Serviceability - AS4100 and NZS3404

Multiframe Steel Codes provides two design checks for the serviceability of a member. These design checks are used to check that the deflection of a member about either the major or minor axes does not exceed a specified deflection limit.

Serviceability Dialog - AS4100 and NZS3404

To set the design properties of a member for serviceability

- **Select the required members in the Frame window**
- **Choose Serviceability … from the Design menu**



- **For each deflection check, select the axis about which the deflection will be checked.**
- **Type in values for the deflection limits.**
- **Click OK**

Seismic (NZS3404)

The design of a member for seismic actions is divided into four design checks and four design constraints. The four design checks consider the axial force limits of clause 12.8.3.1 and the user can choose to check the member for the General Axial Limit (clause 12.8.3.1(a)), Axial Compression Limit for both major and minor axes (clause 12.8.3.1(b)) and the Axial Force Limit (clause 12.8.3.1(c)). The Axial Force Limit is applied using $N^*_g=N^*$.

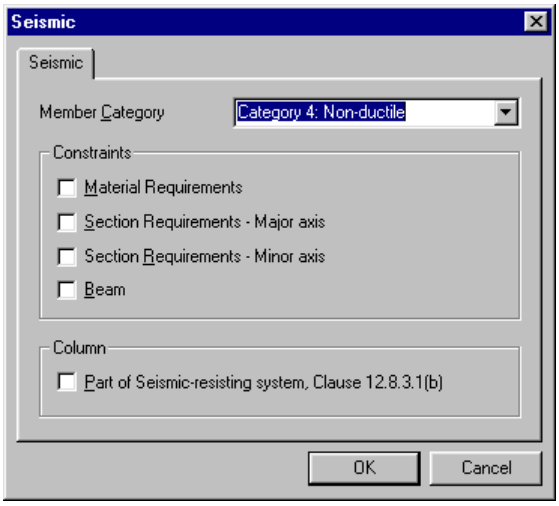
The four design constraints check the member for the Beam, Material and Section Geometry requirements of clauses 12.4.1, 12.5.1 and 12.7.2.1. The user can select which of these constraints are to be applied to the design of a member via the Seismic dialog.

When checking or designing members using NZS3404 it is necessary to specify the category of a member. The category of a member is specified by choosing the appropriate category from the list provided in the Seismic Dialog. The default category for all members is category 4.

NZS3404 Seismic Dialog

To set the seismic design properties of a member

- **Select the required members in the Frame window**
- **Choose Seismic… from the Design menu**



- **Choose the member category from popup menu**
- **Select each of the design constraints to be tested**
- **Identify if the member is part of the seismic resisting system.**
- **Click OK**

Default Design Properties - AS4100 and NZS3404

There are a number of design variables, which are used when doing checking to the code. A summary of all of the design variables is as follows;

Variable	Description	Default
Fy	Yield strength of the section's steel	250Mpa
Fu	Ultimate Tensile Strength of the section's steel	410Mpa
Kx	Effective length factor for buckling about the section's strong axis	1.0
Ky	Effective length factor for buckling about the section's weak axis	1.0
Lcx	Unbraced length for bracing preventing buckling about the section's strong axis	Member's length
Lcy	Unbraced length for bracing preventing buckling about the section's weak axis	Member's length
Lateral restraints	The lateral restraints acting on the member.	Each end of the member is fully restrained at both flanges.
Load Height	The position of the loading on beam (shear centre or top flange).	Shear Centre
s	Spacing of web stiffeners. This is the spacing of any stiffeners along the web of a beam	0.0 (i.e. no stiffeners)
No. of Flange Holes	The number of holes in the flanges of the section.	0
Diameter of Flange Holes	Diameter of holes in the flanges of the section.	0.0

Total Height of Flange Holes	Total height of any boltholes in the flanges of the section. This value may be input directly or computed automatically when the number and diameter of flange holes are specified.	0.0
No. of Web Holes	The number of holes in the webs of the section.	0
Diameter of Web Holes	Diameter of holes in the webs of the section.	0.0
Kt	Correction factor for the distribution of forces.	1.0
Total Height of Web Holes	Total height of any bolt holes in the webs of the section. This value may be input directly or computed automatically when the number and diameter of flange holes are specified.	0.0
Fabrication	The method by which the section was manufactured. This describes the residual stresses in the section.	Rolled
Member Category	Category of member for purposes of seismic design. (NZS3404 only)	4

It is not necessary to enter all of the above information for all members. Usually you will want to check some members for bending, others for compression and so on. The items under the Design menu help you enter just the required information depending on what type of check you are doing.

Code Clauses Checked - AS4100 and NZS3404

When carrying out code checks, Multiframe Steel Codes uses the following clauses of the applicable codes to check your structure. No other checks are performed unless they are specifically listed below.

Checks are not carried out on hybrid members, composite members or tapered members. Checks on mono-symmetric I sections are not considered as are checks using actions computed using plastic analysis.

The alternative design provisions provided by the code for combined actions checks are automatically used if the member meets the required criteria.

AS4100 Clauses Checked

"Australian Standard AS4100-1990: Steel Structures", Standards Australia, October 26, 1990 including Amendment No.1 (August 3, 1992), Amendment No.2 (June 14, 1993) and Amendment No.3 (December 5, 1995).

Clauses used are 4.4, 4.6, 5.1, 5.2, 5.3, 5.6, 5.11, 6.1, 6.2, 6.3, 7.1, 7.2, 7.3, 8.3 and 8.4

The design checking procedure is as follows;

For first order analyses, the design bending moments are amplified using the factors determined using clause 4.4.2 and 4.6.2. Amplification factors for sway frames are not considered and a second order analysis should be used for sway frames requiring moment amplification.

The section is classified as compact, non-compact or slender about its major and minor axes using clause 5.2. The effective area and form factors are determined using clause 6.2.

For major and minor bending section checks, the design bending moment is checked to be less than the nominal section moment design capacity as found using clause 5.2.

For bending member checks, the design bending moment about the major principle axis is checked to be less than the nominal member moment design capacity as found using clauses 5.3 and 5.6. **Clause 5.6.3 and clause 5.6.4 are NOT considered.**

For major and minor shear checks, the design shear force is checked to be less than the nominal shear capacity found from section 5.11. The flange restraint factor (α_f) of clause 5.11.5.2 is always set to 1.0.

For tension checks, the design axial tension force is checked to be less than the nominal section design capacity in tension as computed using clause 7.2.

For compression section checks, the design axial compressive force is checked to be less than the nominal section design capacity in compression as computed using clause 6.2.

For major and minor compression member checks, the design axial compressive force is checked to be less than the nominal member design capacity in compression as computed using clause 6.3. **Clause 6.3.4 is NOT considered.**

For all combined action section checks, the design axial force (N*) is the maximum axial force in the member, and the design bending moments (M_x*, and M_y*) are the maximum bending moments in the member.

For major and minor combined section checks, the design bending moment is checked to be less than the nominal section moment design capacity reduced by axial force (compression or tension) as computed using clause 8.3.2 and 8.3.3.

For combined biaxial section checks, the design bending moments are checked to satisfy clause 8.3.4.

For major and minor combined in-plane member checks, the design bending moment is checked to be less than the nominal in-plane member moment design capacity as computed using clause 8.4.2. **Clause 8.4.3 is NOT considered.**

For combined out-of-plane member checks, the design bending moment about the major axis is checked to be less than the nominal in-plane member moment design capacity as computed using clause 8.4.4.

For combined biaxial member checks, the design bending moments are checked to satisfy clause 8.4.5.

Clause 8.4.6 is NOT considered.

NZS3404 Clauses Checked

"New Zealand Standard NZS3404-1997: Steel Structures", Standards New Zealand, 26th June 1997, **including Draft Amendment No.1 (August, 2000)**.

Clauses used are 4.4, 4.8, 5.1, 5.2, 5.3, 5.6, 5.11, 6.1, 6.2, 6.3, 7.1, 7.2, 7.3, 8.1, 8.3, 8.3, 12.4, 12.5,12.7 and 12.8.

The design checking procedure is as follows;

For first order analyses, the design bending moments are amplified using the factors determined using clause 4.4.2 and 4.8.2. Amplification factors for sway frames are not considered and a second order analysis should be used for sway frames requiring moment amplification.

The section is classified as compact, non-compact or slender about its major and minor axes using clause 5.2. The effective area and form factors are determined using clause 6.2.

The member is checked for compliance to clauses 12.4.1.1, 12.5.1.1 and 12.7.2.1. Compliance of clause 12.4.1.1 only considers the maximum yield stress and the maximum ratio of (f_y/f_u).

For major and minor bending section checks, the design bending moment is checked to be less than the nominal section moment design capacity as found using clause 5.2.

For bending member checks, the design bending moment about the major principle axis is checked to be less than the nominal member moment design capacity as found using clauses 5.3 and 5.6. **Clause 5.6.3 and clause 5.6.4 are NOT considered.**

For major and minor shear checks, the design shear force is checked to be less than the nominal shear capacity found from section 5.11. The flange restraint factor (α_f) of clause 5.11.5.2 is always set to 1.0.

For tension checks, the design axial tension force is checked to be less than the nominal section design capacity in tension as computed using clause 7.2.

For compression section checks, the design axial compressive force is checked to be less than the nominal section design capacity in compression as computed using clause 6.2.

For major and minor compression member checks, the design axial compressive force is checked to be less than the nominal member design capacity in compression as computed using clause 6.3. **Clause 6.3.4 is NOT considered.**

For all combined action section checks, the design axial force (N*) is the maximum axial force in the member, and the design bending moments (M_x*, and M_y*) are the maximum bending moments in the member.

If any combined action checks are to be considered, the member is first checked to determine if it has a significant axial force in accordance with clause 8.1.4. For members without a significant axial force all combined action checks are skipped.

The member is checked to see if the use of alternative design criteria is acceptable. This check is conducted to clause 8.1.5 but does not consider the plate slenderness limits of clause 8.1.5 (b)(i). Hence, alternative design provisions will only be used if the cross section is compact.

For major and minor combined section checks, the design bending moment is checked to be less than the nominal section moment design capacity reduced by axial force (compression or tension) as computed using clause 8.3.2 and 8.3.3.

For combined biaxial section checks, the design bending moments are checked to satisfy clause 8.3.4.

For major and minor combined in-plane member checks, the design bending moment is checked to be less than the nominal in-plane member moment design capacity as computed using clause 8.4.2. **Clause 8.4.3 is NOT considered.**

For combined out-of-plane member checks, the design bending moment about the major axis is checked to be less than the nominal in-plane member moment design capacity as computed using clause 8.4.4.

For combined biaxial member checks, the design bending moments are checked to satisfy clause 8.4.5.

Clause 8.4.6 is NOT considered.

For seismic member checks, the design axial force is checked to satisfy clauses 12.8.3.1(a), (b) and (c). Note that clause 12.8.3.1(c) is checked using $N^*_g = N^*$.

Chapter 5
LRFD

This chapter describes the implementation of the AISC "Load and Resistance Factor Design Specification for Structural Steel Buildings " (LRFD) and "Load and Resistance Factor Design Specification for Single Angle Members" (LRFD SAM) steel design codes within Multiframe Steel Codes. It provides a step-by-step description of how to modify the design properties used by the code.

- Notation
- Design Checks
- Bending
- Tension
- Tension Dialog
- Compression
- Combined Actions
- Serviceability
- Default Design Properties
- Code Clauses Checked

Notation - LFRD

The notation used in Multiframe Steel Codes generally follows that used in the LRFD and LRFD SAM. Use has been made of subscripts to clarify the axis of the member to which a quantity refers. For example, the nominal flexural strengths about the X and Y axes are denoted M_{nx} and M_{ny} respectively.

The geometric axes of a member are denoted as the X and Y axes where X represented the horizontal axis of the member and Y the vertical axis of the member. For design to LRFD, it is assumed that the X axis is the major axis and Y is the minor axis. For most sections these corresponds to the principal axes but for some sections, such as angles, the geometric axes do not correspond to the principal axes. In this case, quantities pertaining to the major and minor principle axes are denoted using U and V respectively.

Design Checks - LFRD

The types of checks are grouped into the categories; Bending, Tension, Compression, Combined and Serviceability. The user may specify which of these checks are performed when a member is designed or checked using Multiframe Steel Codes.

Bending - LFRD

The design of a member for bending is divided into four design checks. These check the flexural and shear capacity of the member about the major and minor axes. Each of these checks may consider one or more limit states depending upon the section and the actions within the member.

When performing a bending check it is necessary to specify how lateral buckling of the member is resisted. Restraint could be provided by other members, purlins, girts or by other structural elements that are not modelled in Multiframe such as concrete slabs. Multiframe Steel Codes provides three methods of specifying how a member is restrained against lateral buckling. The user may specify

That the member is fully restrained against lateral buckling in which case no lateral buckling checks will be performed.
The location and type of lateral restraints applied to the member in which case Multiframe Steel Codes will appropriately divide the member into a number of spans and consider the capacity of each of these spans in determining the capacity of the member.
The laterally unbraced length (L_b) and bending coefficient (C_b).

You may need to specify a number of properties relating to the location and type of lateral restraints and the stiffener spacing along the member

Lateral Restraints - LFRD

If the spacing of lateral restraints along the member is specified, Multiframe Steel Codes uses this information to break the member up into a number of spans in order to determine lateral torsion buckling capacity of each span. In Multiframe Steel Codes, these spans are known as segments.

Each lateral restraint specified by the user is assumed to provide bracing against lateral displacement of the *critical flange* and/or prevent twist of the cross section. At any cross section, the *critical flange* is the flange that, in the absence of any restraint at that cross section, would deflect the furthest during buckling of the member. In most members the *critical flange* will be the compression flange. However for a cantilevered member, the *critical flange* is the tension flange.

For each restraint located along a member, the user must specify the type of restraint. As this depends upon which flange is the critical flange, which is not know *a priori*, the user must specify the type of lateral restraint that would be present at a section if

- **The top flange was the critical flange, and**
- **The bottom flange was the critical flange.**

In LRFD no distinction is made between different types of lateral restraints. However, to be compatible with other design codes, Multiframe Steel Codes allows for lateral restraints at a cross section to be classified as follows

- **Full Restraint –supports the cross section against lateral displacement of the critical flange and prevents twist of the cross section.**
- **Partial Restraint – provides support against lateral displacement of the section at a point other than the critical flange and prevents twist of the cross section.**
- **Lateral Restraint – resists lateral displacement of the critical flange only.**

For the purpose of design in LRFD, each of these restraint types is consider adequate to provide lateral support to the cross section at which they are applied.

Lateral restraints must always be specified at the ends of the beam and so the minimum number of lateral restraints is two. If no restraint exists at the end of a member then it should be specified as unrestrained in which case the member would be regarded as a cantilever. The initial lateral restraints applied to the member are full restraints at each end for either of the flanges being the critical flange.

The location and type of lateral restraints can be displayed in the Frame and Plot windows. The display of lateral restraints can be turned on or off via the Symbols Dialog which contains options for displaying and labelling lateral restraints. The restraints are drawn as a short line in the plane of the major axis of the member. These lines extend each side of the member for a distance that is roughly the scale of a purlin or girt. Lateral restraints are also displayed in the rendered view of the frame in which they are draw to extend from each flange by approximately the size of a purlin. The restraints may be labelled using a one or two letters to indicate the type of restraint (e.g. F - fixed, P – partial, L - lateral).

Note that lateral restraints at the end of a member are draw slightly offset from the node so that restraints at the ends of connected members may be more readily distinguished.

Unbraced Length (L_b) and Bending Coefficient (C_b) - LFRD

Instead of specifying the position of lateral restraints it may be preferable to directly set the laterally unbraced length of the member. When doing this, it is also necessary to specify the bending coefficient (C_b) as this can no longer be automatically determined by Multiframe Steel Codes. LRFD permits a conservative value of C_b=1.0 to be adopted which is the default value used by Multiframe Steel Codes.

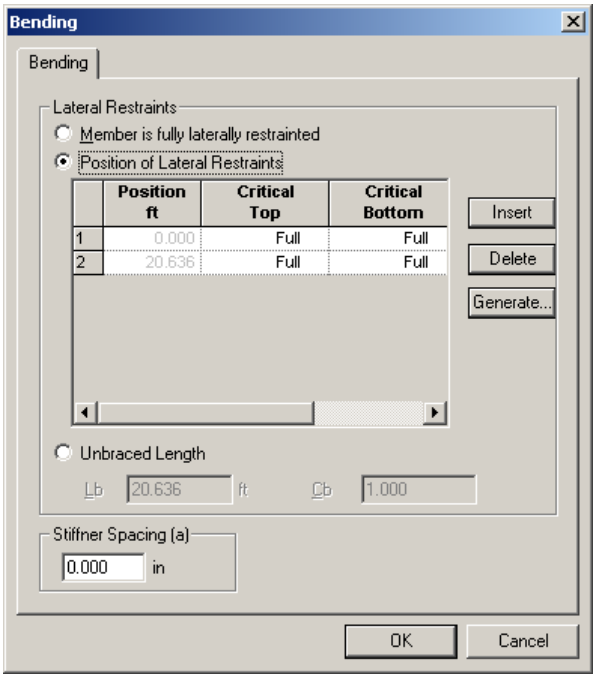
Web Stiffener Spacing - LFRD

When checking or designing a member for bending, you may need to specify the spacing of any stiffeners along the web of the member. This affects the member's susceptibility to buckling due to bending. If there are no transverse stiffeners, you should leave the stiffener spacing set to zero.

Bending Dialog - LFRD

To set the properties for bending

- **Select the required members in the Frame window**
- **Choose Bending from the Design menu**



- **Select the "Member is fully laterally restrained" option, or**
- **Select the "Position of Lateral Restraints" option, and then**

To add new restraint to the member

- **Position the cursor with the table and click the Insert button to add a lateral restraint to the member.**
- **Select the position of each restraint**
- **Select the type of each lateral restraint from the combo provided in each cell.**

or

- **Click the Generate button to automatically generate a number of restraints.**

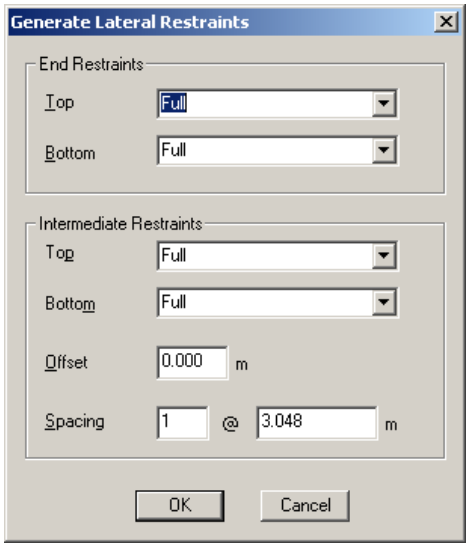
To delete a restraint from the member

- **Position the cursor within the table on the lateral restraint to be deleted and click the Delete button.**
- **Or select the "Unbraced Length" option, and then**
- **Enter the unbraced length (l_e)**
- **Enter the moment modification factor coefficient (α_m) to be used in the design of this length of the member.**
- **Choose the position of the load from popup menu**
- **If there are transverse stiffeners on the web, type in values for the stiffener spacing (s)**
- **Click OK**

Generate Lateral Restraints Dialog - LFRD

When the user selects to generate the lateral restraints from the Bending dialog, the Generate Lateral Restraints dialog is displayed. This dialog enables the user to generate lateral restraints are a specified spacing along the member.

- **From the Bending dialog, click the Generate… button**



- **Select the type of restraints to be used at the ends of the member**
- **Select the type of restraints to be used at intermediate points within the member**
- **Enter the offset length at which the first intermediate restraint will be positioned. Leave this field as zero if no offset is same as the spacing**
- **Enter the number and size of spacings for the intermediate restraints.**
- **Click OK**

All lateral restraint applied to the member will now be regenerated and will replace all existing restraints.

Tension - LFRD

The capacity of a member to resist tensile forces is implemented as a single design check. A number of modification factors may be entered to change the section properties used for checking tension. This includes the area of holes in the flange or web of the member and an area reduction factor to account for the distribution of forces at the ends of a member.

In addition to checking the tensile capacity of the member, a design constraint will be applied to the member enforcing the slenderness of the member to be less than 300.

Bolt Holes - LFRD

When checking or designing a member for tension, you need to specify any reduction in area due to boltholes or other openings within the section. If the members contain significant areas of boltholes, which need to be taken into account when determining the cross-sectional area of the section, you will need to enter the amount of cross-sectional area to be deducted to allow for these holes. The net area of the section is the gross area minus the combined area of boltholes in the flange and web.

The reduction in area can be specified by setting the number and diameter of holes in the web or flanges or the member. Alternative, the user may override this and directly specify the height of holes across the flanges and webs of the cross section. These heights are multiplied by the thickness of the section to determine the total reduction in area of the section. The initial value for the area of boltholes is zero.

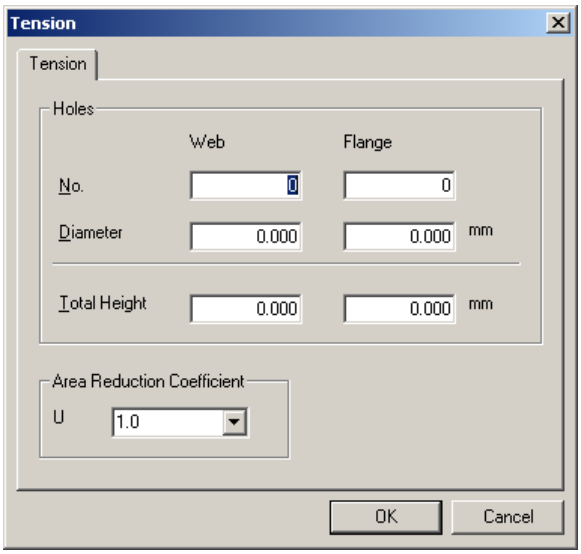
Reduction Coefficient - LFRD

When checking or designing a member for tension using LRFD, you need to specify the reduction coefficient for the distribution of forces at the ends of the member. This coefficient is used to factor the net area in order to compute the effective area. The reduction coefficient U has a default value of 1.0

Tension Dialog - LFRD

To enter the properties for tension

- **Select the required members in the Frame window**
- **Choose Tension… from the Design menu**



- **Type in the number and diameter of holes in the webs and flanges (and the total height of holes will be computed automatically) or…**
- **Type the total height of holes in the webs and flanges directly**
- **Choose or enter a value for the reduction coefficient (U)**
- **Click OK**

Compression - LFRD

Multiframe Steel Codes splits the compressive design of a member to LRFD into two design checks. You may choose to check the member capacity and/or the member's slenderness about the major and minor axes.

When checking or designing members for compression, it is necessary to specify the effective length factors and unbraced lengths of the member.

To determine the critical buckling condition of a member, it is also necessary to know the spacing of any bracing (if any) along the member. This bracing could be provided by purlins, girts or other structural elements, which are not modelled in Multiframe. Some bracing may only restrain lateral deflection in one direction, therefore it is necessary to enter unbraced lengths for both axes of the section, Lcx corresponding to the spacing of restraints preventing compression buckling about the x-x axis and Lcy corresponding to the spacing of restraints preventing compression buckling about the y-y axis.

To determine the critical buckling load for a member, it is necessary to enter an effective length to indicate the type of restraint on the ends of the member. The effective length is given by an effective length factor multiplied by the unbraced length of the member. The effective length may be different for buckling in the major and minor axis directions. The effective lengths are given by

Lx = Kx * Lcx , Ly = Ky * Lcy and Lz = Kz * Lcz

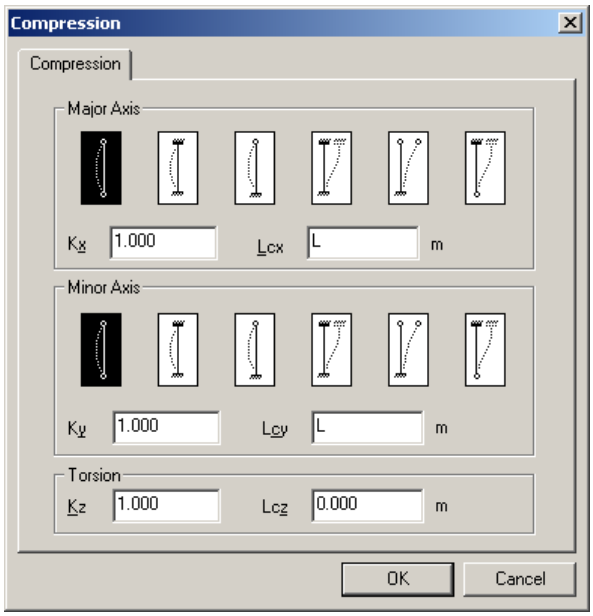
Where Lcx and Lcy is the unbraced length of the member and Kx, Ky the two effective length factors for the major and minor axes respectively. Lcz is the unbraced length and Kz is the effective length factor of the member for torsional buckling. The initial values of Kx, Ky and Kz are 1.0 and the initial values of Lcx, Lcy and Lcz are the length of the member.

In addition to checking the compressive capacity of the member, a design constraint will be applied to the member enforcing the slenderness of the member to be less than 200.

Compression Dialog - LFRD

To set the properties for compression

- **Select the required members in the Frame window**
- **Choose Compression… from the Design menu**



Either

- **Click on the icons for the end conditions in each direction or…**
- **Type in values for Kx and Ky**
- **Type in values for Lcx and Lcy**
- **Type in values for Kz and Lcz**
- **Click OK**

If you choose a standard end condition, the recommended Kx and Ky values will be automatically entered for you.

Combined Actions - LFRD

The design of a member for combined actions is divided into three design checks. The user can select to check the member for biaxial bending or biaxial bending in conjunction with either a tensile or compressive axial force. The user is not required to provide any additional design properties for the combined actions checks as it uses results already derived from the tension, compression and bending checks.

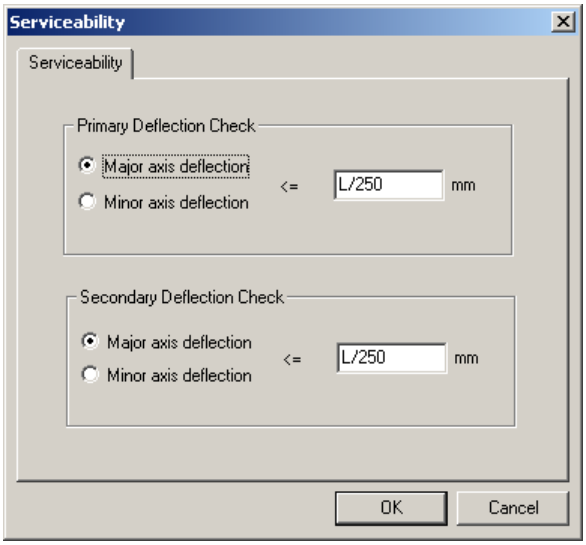
Serviceability - LFRD

Multiframe Steel Codes provides two design checks for the serviceability of a member. These design checks are used to check that the deflection of a member about either the major or minor axes does not exceed a specified deflection limit.

Serviceability Dialog - LFRD

To set the design properties of a member for serviceability

- **Select the required members in the Frame window**
- **Choose Serviceability … from the Design menu**



- **For each deflection check, select the axis about which the deflection will be checked.**
- **Type in values for the deflection limits.**
- **Click OK**

Default Design Properties - LFRD

There are a number of design variables, which are used when doing checking to the code. A summary of all of the design variables is as follows;

Variable	Description	Default
Fy	Yield strength of the section's steel	250Mpa
Fu	Ultimate Tensile Strength of the section's steel	410Mpa
Kx	Effective length factor for buckling about the section's strong axis	1.0
Ky	Effective length factor for buckling about the section's weak axis	1.0
Kz	Effective length factor for torsional buckling.	1.0
Lcx	Unbraced length for bracing preventing buckling about the section's strong axis	Member's length
Lcy	Unbraced length for bracing preventing buckling about the section's weak axis	Member's length
Lcy	Unrestrained length for bracing preventing torsional buckling	Member's length
Lateral restraints	The lateral restraints acting on the member.	Each end of the member is fully restrained at both flanges.
Lb	Unrestrained length of member for lateral torsional buckling.	Member's length
Cb	Bending coefficient.	1.0
s	Spacing of web stiffeners. This is the spacing of any stiffeners along the web of a beam	0.0 (i.e. no stiffeners)
No. of Flange Holes	The number of holes in the flanges of the section.	0
Diameter of Flange Holes	Diameter of holes in the flanges of the section.	0.0
Total Height of Flange Holes	Total height of any boltholes in the flanges of the section. This value may be input directly or computed automatically when the number and diameter of flange holes are specified.	0.0
No. of Web Holes	The number of holes in the webs of the section.	0
Diameter of Web Holes	Diameter of holes in the webs of the section.	0.0
U	Correction factor for the distribution of forces.	1.0
Total Height of Web Holes	Total height of any boltholes in the webs of the section. This value may be input directly or computed automatically when the number and diameter of flange holes are specified.	0.0
Fabrication	The method by which the section was manufactured. This describes the residual stresses in the section.	Hot Rolled

It is not necessary to enter all of the above information for all members. Usually you will want to check some members for bending, others for compression and so on. The items under the Design menu help you enter just the required information depending on what type of check you are doing.

Code Clauses Checked - LFRD

When carrying out code checks, Multiframe Steel Codes uses the following clauses of the applicable codes to check your structure. No other checks are performed unless they are specifically listed below.

Checks are not carried out on composite members or tapered members. Checks on mono-symmetric I sections are not considered as are checks using actions computed using plastic analysis.

- **LRFD**
- **LRFD SAM**

LRFD Clauses Checked

"Load and Resistance Factor Design Specification for Structural Steel Buildings", American Institute of Steel Construction, December 27, 1999.

The design checking procedure is as follows:

The net area of the section is computed by subtracting the area of holes in the section. The effective area is then calculated as the net area (A_n) times the area reduction coefficient (U).

If the member is been checked for tension of compression, the slenderness of the section is checked to ensure that it meets the limits set out in Section B7. For angle members, the slenderness about either of the geometric axes is determined using the minimum radius of gyration of the section.

If the member is a plate web girder, the section is checked to determine is if meets the web slenderness limits specified in Appendix G1.

For each serviceability load case:

The maximum local displacement of the member is compared to the deflection limits specified deflection limits.

For each load case representing a strength limit state,

The design actions, or required strengths, of the member are determined as the maximum moment, shears and axial forces within the member.

For first order analyses, the design bending moments are amplified using the factors determined using clause C2. Only moment amplification of braced frames is considered which corresponds to the situation in which no moments result from the lateral translation of the frame. As such, moment amplification is computed using only the first term of the right hand side of equation C1-1. Amplification factors for sway frames are not considered and a second order analysis should be used for sway frames requiring moment amplification.

The plate elements of the section will be classified as Compact/Non-Compact/Slender as per the requirements of clause B5.1 and Table B5-1. These elements may also be classified as Very Slender if they exceed the limitations set out in Table A-F1.1. If the moments in the member are less than one ten thousandth of the yield moments the section is considered to be in pure compression and will be classified accordingly. If an element of the section is found to be slender, the stiffness reduction factors Q, Q_a and Q_s will be determined as set out in Appendix B.

For tension checks, the capacity of the member is determined in accordance with section D1.

For compression checks, the capacity of the member is firstly computed for the limit states of flexural buckling about the major and minor axis is accordance with clause E2. The capacity of the member for the limit state of flexural torsional-buckling is then computed using clauses E3 and Appendix E. The compressive capacity of the member is regarded as being the minimum capacity determined for these three limit states.

For bending checks the provisions of Appendix F1 are used. For each of the failure modes, yielding, flange local buckling, web local buckling and lateral torsional buckling, λ, λ_p and λ_r values are calculated. The values are based upon the section shape and the axis of bending and are derived from Table A-F1.1. After the various λ values have been calculated they are then compared to find the appropriate equation to calculate M_n, Equ. A-F1-1 to 4. Each M_n value for the failure modes are then compared with the lowest value governing.

Flange local bucking will only be considered for sections with non-compact flanges. Similarly, web local buckling will only be considered for sections with non-compact webs.

The design for shear is carried out in accordance with clause F2 using the provisions of Appendix F2.2 when a stiffener spacing is specified. For plate girders with slender web elements, the provisions of Appendix G3 will be utilised instead. **No calculations are conducted using Chapters K or J.**

For the biaxial bending check, interaction equations of Appendix H1 are evaluated ignoring the axial force term. The expressions are computed using the **maximum** actions in the members. If this check fails, the user

For the combined action check for flexure and compression, the member is checked in accordance with clause H1.1 using the design moments about the major and minor axes. A more refined

LRFD SAM Clauses Checked

"Load and Resistance Factor Design Specification for Single Angle Members", American Institute of Steel Construction, November 10, 2000.

The design checking procedure is the same as described above for LRFD except that:

The section is classified using the limits set out in clause 4 of LRFD SAM. The same clause is used to compute the slenderness reduction factors and effective area of the section.

Clause 2 of LRFD SAM is used to determine the tensile capacity of the member.

For the bending checks, the shear is determined using clause 3 of LRFD SAM while the flexural capacity is determined using clause 5 of LRFD SAM.

The lateral-torsional buckling capacity of the member for the limit state of lateral-torsion buckling of unequal angle sections without lateral torsion restraint or sections modelled about their principle is not yet supported. When such a section is encountered, the member will have determined to have no flexural capacity.

The capacity of a member under combined forces is computed using clause 6 of LRFD SAM in place of the provisions in clause H or LRFD.

Chapter 6
BS5950

This chapter describes the implementation of the British BS5950 steel design code within Multiframe Steel Codes. It provides a step-by-step description of how to modify the design properties used by the code.

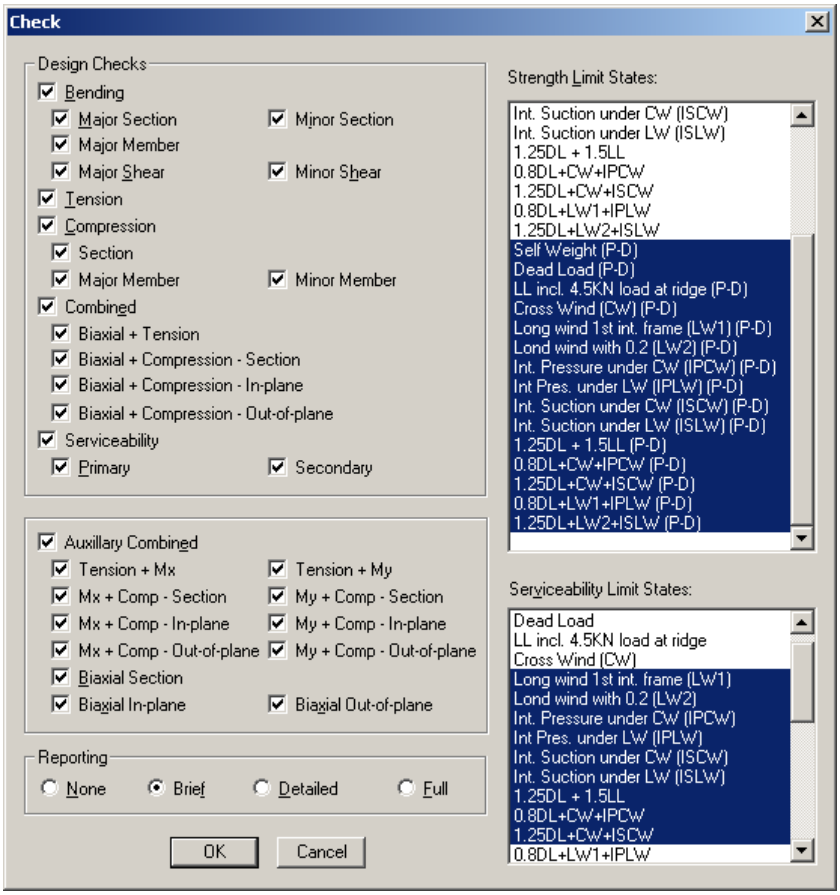
- Notation
- Design Checks
- Bending
- Tension
- Compression
- Combined Actions
- Serviceability
- Default Design Properties
- Code Clauses Checked

Notation - BS5950

The notation used in Multiframe Steel Codes generally follows that used in BS5950.

Design Checks - BS5950

The types of checks are grouped into the categories; Bending, Tension, Compression, Combined and Serviceability. In addition, a number of auxiliary combined action checks have been included that consider axial force and bending about a single axis only. The user may specify which of these checks are performed when a member is designed or checked using Multiframe Steel Codes.



Bending - BS5950

The design of a member for bending consists of five design checks. These check the section capacity of the member about the major and minor axes, the shear capacity about both axes and the member, and the buckling, capacity about the major axis.

When performing a bending check it is necessary to specify how lateral-torsional buckling of the member is resisted. Restraint could be provided by other members, purlins, girts or by other structural elements that are not modelled in Multiframe such as concrete slabs. Multiframe Steel Codes provides three methods of specifying how a member is restrained against lateral buckling. The user may specify

- **That the member is fully restrained against lateral buckling in which case no lateral buckling checks will be performed, or**
- **The location and type of lateral and torsional restraints applied to the member in which case Multiframe Steel Codes will appropriately divide the member into a number of spans and consider the capacity of each of these spans in determining the capacity of the member, or**
- **The laterally unbraced length (L_b) and moment modification factor (m_{LT}).**

You may also need to specify a number of properties relating to the location and type of lateral restraints and the stiffener spacing along the member

Lateral and Torsional Restraints - BS5950

To compute the buckling capacity of a member it is necessary to know the spacing of any lateral and torsional restraints (if any) along the member. The restraints could be provided by purlins, girts or other structural elements, which are not modelled in Multiframe. Multiframe Steel Codes uses this information to determine the length of segments used in the design calculations for lateral torsional buckling. In Multiframe Steel Codes, The restraint provided by a support is described by how it restraints the top and bottom flanges and how it restraints the cross-section of the member at that location against torsion.

Restraints must always be specified at the ends of the member. If no actual restraint exists at the end of a member then it should be specified as unrestrained. Lateral restraints at the ends of a member may also be specified as providing either full or partial restraint against rotation on plan. By default, the ends of a member will be assumed to be laterally restraint at both the top and bottom flange but provide no resistance to on plan rotation of the member. Torsional restraints at the ends of a member may be specified as unrestrained, fully restrained, partially restrained or frictionally restrained. Partial restraints inhibit the rotation of the cross section by the connection of the bottom flange to the supports while frictional restraints resist rotation of the member about its longitudinal axis by only the pressure of the bottom flange onto its supports (Refer to Table 13 of BS5950).

Intermediate restraints applied to the member may provide lateral and torsional restraint. No distinction is made for the on-plan rotational resistance that may be provided by lateral restraints.

The location and type of lateral restraints can be displayed in the Frame and Plot windows. The display of lateral restraints can be turned on or off via the Symbols Dialog which now contains options for displaying lateral restraints and labelling these restraints.

The restraints are draw as a short line in the plane of the major axis of the member. These lines extend each side of the member for a distance that is roughly the scale of a purlin or girt. Lateral restraints are also displayed in the rendered view of the frame in which they are draw to extend from each flange by approximately the size of a purlin. The restraints may be labelled using a one or two letters to indicate the type of restraint. Lateral are labelled using the following notation

U – Unrestrained
L – Lateral restraint
LR – Lateral restraint with full restraint against rotation on plan
LP – Lateral restraint with partial restraint against rotation on plan

Note that lateral restraints at the end of a member are draw slightly offset from the node so that restraints at the ends of connected members may be more readily distinguished.

Unbraced Length (L_b) and Bending Coefficient (m_{LT}) - BS5950

Instead of specifying the position of lateral restraints it may be preferable to directly set the laterally unbraced length of the member. When doing this, it is also necessary to specify the bending coefficient (m_{LT}) as this can no longer be automatically determined by Multiframe Steel Codes. The design codes permit a conservative value of m_{LT} =1.0 to be adopted which is the default value used by Multiframe Steel Codes.

Web Stiffener Spacing - BS5950

When checking or designing a member for bending, you may need to specify the spacing of any stiffeners along the web of the member. This affects the member's susceptibility to buckling due to bending. If there are no transverse stiffeners, you should leave the stiffener spacing set to zero.

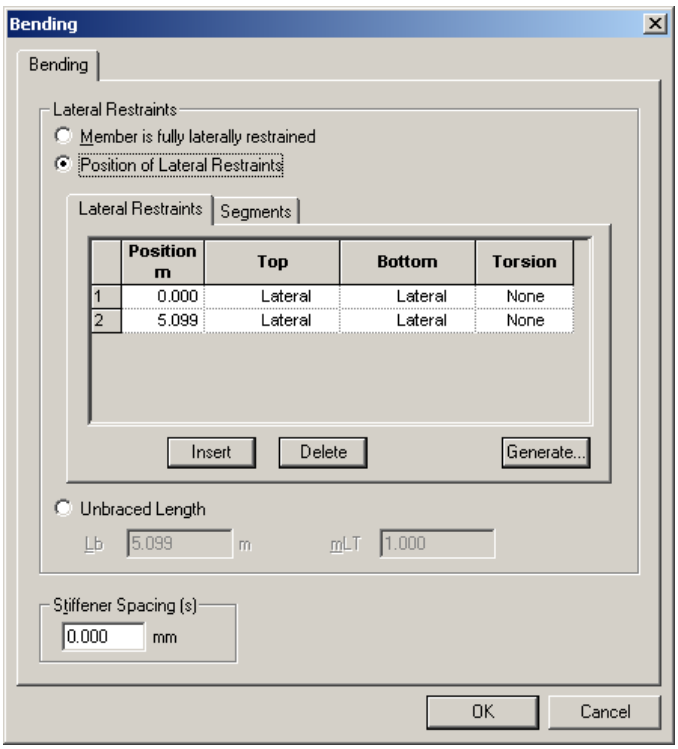
Load Height - BS5950

When checking or designing a member for bending, you may need to specify the load height position. This is used in determining the effective lengths of segments or sub-segments along the member.

Bending Dialog - BS5950

To set the properties for bending

- **Select the required members in the Frame window**
- **Choose Bending from the Design menu**



If the member is fully braced against lateral torsion buckling

- **Select the "Member is fully laterally restrained" option**

or if the location of lateral bracing along the member is to be specified

- **Select the "Position of Lateral Restraints" option**

To add new restraint to the member

- **Position the cursor with the table and click the Insert button to add a lateral restraint to the member.**
- **Select the position of each restraint**

- **Select the type of each lateral restraint from the combo provided in each cell.**

or

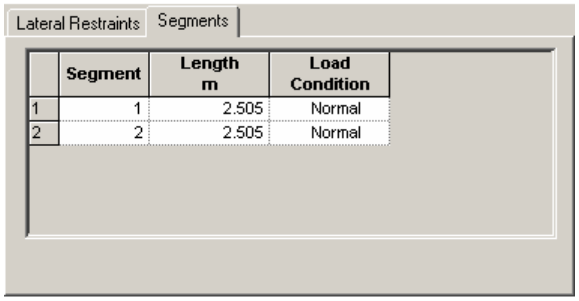
- **Click the Generate button to automatically generate a number of restraints.**

To delete a restraint from the member

- **Position the cursor within the table on the lateral restraint to be deleted and click the Delete button.**

And then to display the list so segment defined by the restraints

- **Click on the Segments tab**



- **For each segment choose the position of the load from popup menu**

or if the unbraced length of the member if the be specified directly

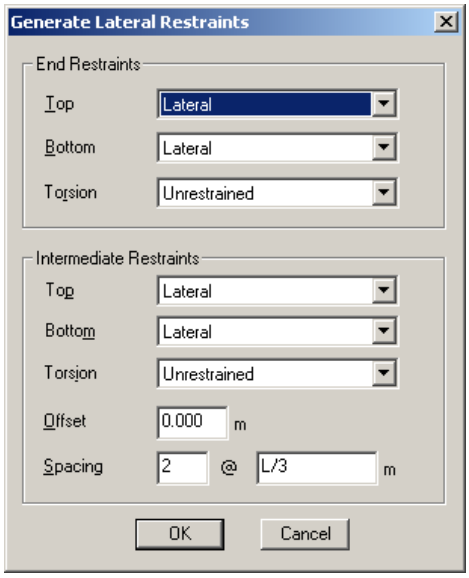
- **Select the "Unbraced Length" option**
- **Enter the unbraced length (l_e)**
- **Enter the moment modification factor coefficient (m_{LT}) to be used in the design of this length of the member.**
- **If there are transverse stiffeners on the web, type in values for the stiffener spacing (s)**
- **Click OK**

Generate Lateral Restraints Dialog - BS5950

When the user selects to generate the lateral restraints from the Bending dialog, the Generate Lateral Restraints dialog is displayed. This dialog enables the user to generate lateral restraints at a specified spacing along the member.

- **From the Bending dialog, click the Generate… button**

The Generate Lateral Restrains dialog will appear allowing you to specify the restraints to be generated.



- **Select the type of restraints to be used at the ends of the member**
- **Select the type of restraints to be used at intermediate points within the member**
- **Enter the offset length at which the first intermediate restraint will be positioned. Leave this field as zero if offset is the same as the spacing**
- **Enter the number and spacing between the intermediate restraints.**
- **Click OK**

All lateral restraint applied to the member will now be regenerated and will replace all existing restraints.

Tension - BS5950

The capacity of a member to resist tensile forces is implemented as a single design check. A number of modification factors may be entered to change the section properties used for checking tension. This includes the area of holes in the flange or web of the member and a correction factor to account for the distribution of forces at the ends of a member.

Bolt Holes - BS5950

When checking or designing a member for tension, you need to specify any reduction in area due to boltholes or other openings within the section. If the members contain significant areas of boltholes, which need to be taken into account when determining the cross-sectional area of the section, you will need to enter the amount of cross-sectional area to be deducted to allow for these holes. The net area of the section is the gross area minus the combined area of boltholes in the flange and web.

The reduction in area can be specified by setting the number and diameter of holes in the web or flanges or the member. Alternative, the user may override this and directly specify the height of holes across the flanges and webs of the cross section. These heights are multiplied by the thickness of the section to determine the total reduction in area of the section. The initial value for the area of boltholes is zero.

Area Reduction Coefficient - BS5950

The reduced tensile capacity of members with eccentric connections is specified by clause 4.6.3 of BS5950. Multiframe Steel Codes does not use this clause but instead approximates the tensile capacity using a similar calculation to that specified by Clause 4.6.1 but which includes an extra factor to account for the reduction in area. As such that the tensile capacity is computed in Multiframe Steel Codes using the expression

$P_t = p_y k_t A_e$

in which k_t represents an area reduction coefficient.

While this method does not directly represent the calculation of clause 4.6.3.1 it provides a simple method by which to account for the reduced tensile capacity described in this clause. For the tensile capacity expressions of clause 4.6.3 is can be shown that minimum values of k_t are

Clause 4.6.3.1 – bolted connections $P_t = p_y(A_e-0.5a_2) \rightarrow$ **kt = 0.5**
– welded connections $P_t = p_y(A_g-0.3a_2) \rightarrow$ **kt = 0.7**

Clause 4.6.3.2 – bolted connections $P_t = p_y(A_e-0.25a_2) \rightarrow$ **kt = 0.75**
– welded connections $P_t = p_y(A_e-0.15a_2) \rightarrow$ **kt = 0.85**

while less conservative values of k_t based upon *the gross area of the connected element taken as half the gross are of the section* are as follows.

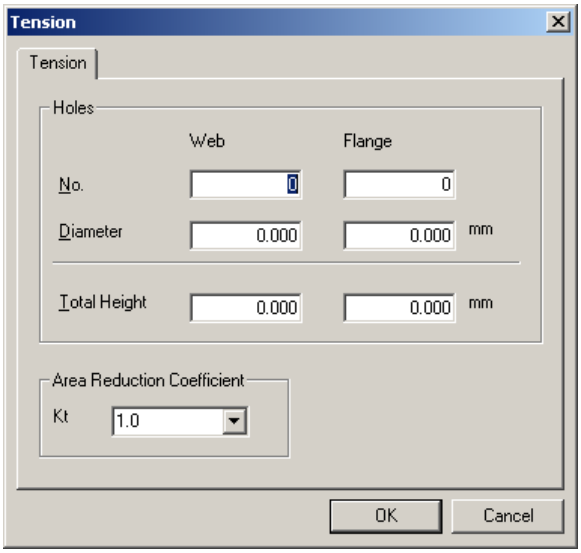
Clause 4.6.3.1 – bolted connections $P_t = p_y(A_e-0.5a_2) \rightarrow$ **kt = 0.75**
– welded connections $P_t = p_y(A_e-0.3a_2) \rightarrow$ **kt = 0.85**

Clause 4.6.3.2 – bolted connections $P_t = p_y(A_e-0.25a_2) \rightarrow$ **kt = 0.875**
– welded connections $P_t = p_y(A_e-0.15a_2) \rightarrow$ **kt = 0.925**

Tension Dialog - BS5950

To enter the properties for tension

- **Select the required members in the Frame window**
- **Choose Tension… from the Design menu**



- **Type in the number and diameter of holes in the webs and flanges (and the total height of holes will be computed automatically) or…**
- **Type the total height of holes in the webs and flanges directly**
- **Choose or enter a value for the Area Reduction Coefficient (kt) if required**
- **Click OK**

Compression - BS5950

Multiframe Steel Codes splits the compressive design of a member to BS5950 into three design checks. You may choose to check the section capacity and/or the member buckling capacities about the major and minor axes.

The section capacity check calculates the capacity of the members cross-section to carry the axial load and computes the capacity of the members as simply the gross area times the yield strength. This check is not explicitly defined in BS5950 as the capacity of the cross section will always be adequate if the member satisfies the member buckling checks. However, this check has been provided within Multiframe Steel Codes to help distinguish this type of failure mechanism in the design of the column.

To determine the buckling capacity for a column it is necessary to know the spacing of any bracing (if any) along the member. This bracing could be provided by purlins, girts or other structural elements, which are not modelled in Multiframe. Some bracing may only restrain lateral deflection in one direction therefore it is necessary to enter unbraced lengths for both axes of the section. In Steel Design the unbraced length of a member may be specified in either of the following ways;
By specifying a single unbraced length and effective length factor for buckling about each axis, or
By breaking the member into column segments and setting the effective length factor for each segment. Each column segment is then designed separately for compression.

Unbraced Lengths and Effective Length Factors - BS5950

To determine the buckling load for a member the user may choose to specify a single unbraced length of the member for buckling about each principle axis. It is also necessary to enter an effective length factor to indicate the type of restraint applied to the ends of the unbraced span of the column. These may be different for buckling in the major and minor axis directions. The effective lengths for determining the buckling capacity of the member are given by

Lx=Kx*Lcx and Ly=Ky*Lcy

where Lcx and Lcy are the unbraced lengths of the member and Kx and Ky are the two effective length factors for the major and minor axes respectively.

The initial values of Lcx and Lcy are the length of the member and the initial values of Kx and Ky are 1.0.

Column Segments - BS5950

A more sophisticated method for the design of a member for compression allows for the division of the member into a number of column segments. These segments are defined by restraints that resist column buckling that are applied at intervals along the member. **In Multiframe Steel Codes, restraints against buckling can be specified at joints along a design member**. These restraints are used to break the member into a number of column segments that may differ for the design of the member about its major and minor axis. The effective length associated with each segment may also be specified to account for the restraint conditions at each ends of the segment.

When column segments are specified, the design of the member will be performed by considering the design of each segment separately.

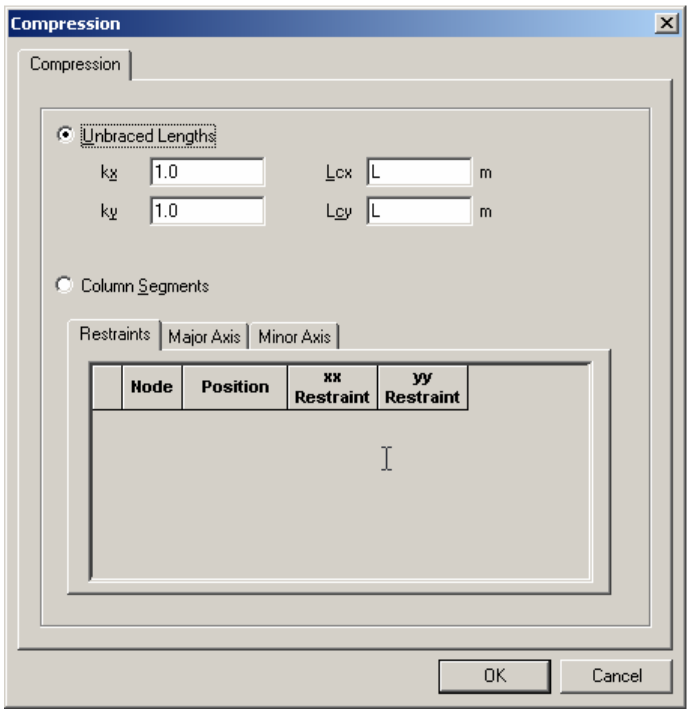
Compression Dialog - BS5950

To set the properties for compression

- **Select the required members in the Frame window**
- **Choose Compression… from the Design menu**

If the unbraced lengths of the member are to be specified directly then

- **Select the Unbrace Length radio button.**

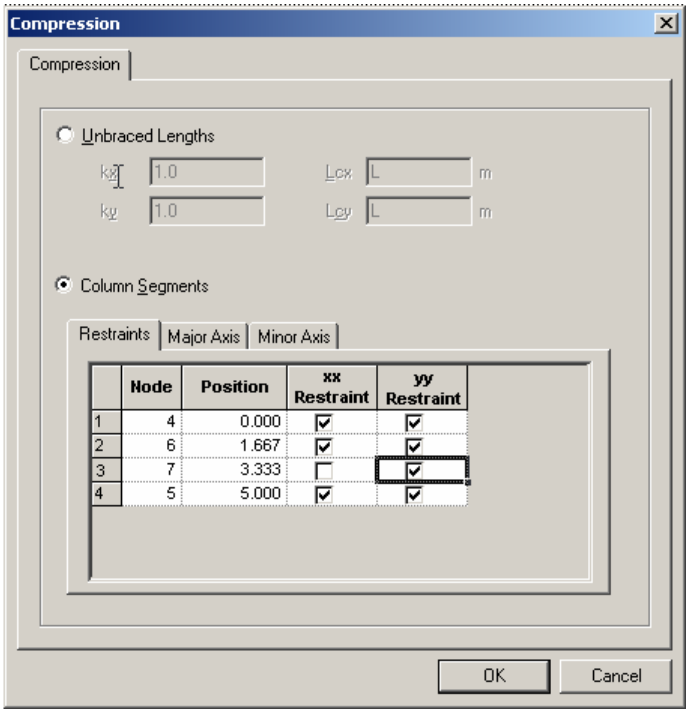


- **Type in values for Kx and Ky**
- **Type in values for Lcx and Lcy**
- **Click OK**

Otherwise if the design for compression is to be performed using column segments.

- **Select the Column Segments radio button.**

The tabbed control in the dialog will become active. The first page in this table lists the location of joints along the members and indicates if they provide restraint against column bucking about either axis of the member.

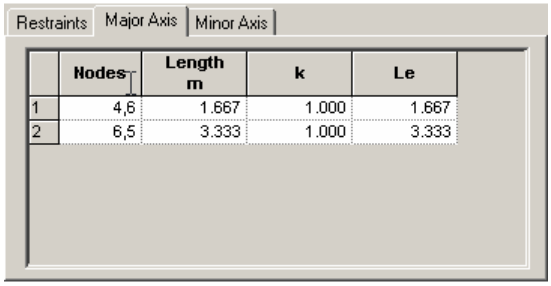


- **Enter the restraints associated with each node.**

The restraint information is used to build a list of column segments that span between the specified restraints.

- **Click on the Major Axis tab.**

This displays a table of column segments that will be used for the design of the member for compression when considering buckling about the major axis.



- **Enter the effective length factor (K) for each segment.**

- **Click on the Minor Axis tab and enter the effective length factors for the minor axis column segments.**

- **Click OK.**

Combined Actions - BS5950

The design of a member for combined actions is divided into four design checks. The user can select to check the capacity of the member for biaxial bending combined with axial tension and or axial compression. The combined bending and axial compression check is split into three separate calculations, these determine the capacity of the member based upon in-plane bucking, out-of-plane buckling and section failure.

In addition to the four main combined action checks, 11 auxiliary design checks may be considered. These checks determine the capacity of the member using various combinations of two combined actions. These include checks for biaxial bending (no axial force), axial tension or compression combined with bending about the major or minor axis.

No design properties are required when checking or designing members for combined actions using BS5950.

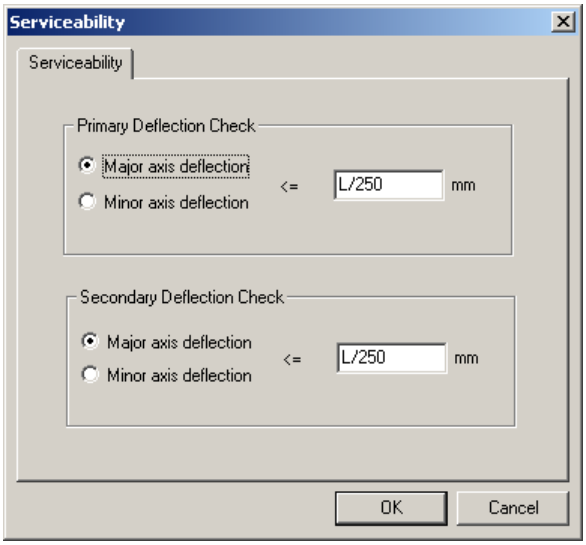
Serviceability - BS5950

Multiframe Steel Codes provides two design checks for the serviceability of a member. These design checks are used to check that the deflection of a member about either the major or minor axes does not exceed a specified deflection limit.

Serviceability Dialog - BS5950

To set the design properties of a member for serviceability

- **Select the required members in the Frame window**
- **Choose Serviceability … from the Design menu**



- **For each deflection check, select the axis about which the deflection will be checked.**
- **Type in values for the deflection limits.**
- **Click OK**

Default Design Properties - BS5950

There are a number of design variables, which are used when doing checking to the code. A summary of all of the design variables is as follows;

Variable	Description	Default
py	Design strength of the section's steel	235Mpa
Us	Minimum Tensile Strength of the section's steel	340Mpa

Kx	Effective length factor for buckling about the section's strong axis	1.0
Ky	Effective length factor for buckling about the section's weak axis	1.0
Lcx	Unbraced length for bracing preventing buckling about the section's strong axis	Member’s length
Lcy	Unbraced length for bracing preventing buckling about the section's weak axis	Member’s length
Lateral restraints	The lateral restraints acting on the member.	Each end of the member is fully laterally restrained at both flanges.
Lb	Unbraced length for lateral torsional buckling	Member’s length
mLT	Equivalent uniform moment factor for lateral torsional buckling	1.0
Load Height	The position of the loading on beam (shear centre or top flange).	Shear Centre
s	Spacing of web stiffeners. This is the spacing of any stiffeners along the web of a beam	0.0 (i.e. no stiffeners)
No. of Flange Holes	The number of holes in the flanges of the section.	0
Diameter of Flange Holes	Diameter of holes in the flanges of the section.	0.0
Total Height of Flange Holes	Total height of any boltholes in the flanges of the section. This value may be input directly or computed automatically when the number and diameter of flange holes are specified.	0.0
No. of Web Holes	The number of holes in the webs of the section.	0
Diameter of Web Holes	Diameter of holes in the webs of the section.	0.0
Kt	Correction factor for the distribution of forces.	1.0
Total Height of Web Holes	Total height of any bolt holes in the webs of the section. This value may be input directly or computed automatically when the number and diameter of flange holes are specified.	0.0
Fabrication	The method by which the section was manufactured. This describes the residual stresses in the section.	Rolled

It is not necessary to enter all of the above information for all members. Usually you will want to check some members for bending, others for compression and so on. The items under the Design menu help you enter just the required information depending on what type of check you are doing.

Code Clauses Checked - BS5950

When carrying out code checks, Multiframe Steel Codes uses the following clauses of the applicable codes to check your structure. No other checks are performed unless they are specifically listed below.

The alternative design provisions provided by the code for combined actions checks are automatically used if the member meets the required criteria.

BS5950

"British Standard BS5950-1:2000: Structural use of steelwork in buildings – Part 1", British Standards Institution, May 15, 2000.

Clauses used 3.4, 3.5, 3.6, 4.2, 4.3, 4.4, 4.6, 4.7, and 4.8. Reference is also made to Annex's B.2, C1, C.2, I.2 and I.3.

The design checking procedure is as follows;

Any section properties missing from the sections library that are required for the design of the section are computed.

The section is classified as plastic, compact, non-compact or slender using clause 3.5.2. **Any section shape not supported by Multiframe Steel Codes shall be classified as compact**.

For sections classified as class 3 semi-compact, the effective plastic moduli are computed using clause 3.5.6.

For sections classified as class 4 slender, the effective area and effective elastic moduli are computed using clause 3.6. **Only the design of symmetric I sections with slender flanges, rectangular hollow sections, equal angles and circular hollow sections are supported by this design module.**

For major and minor shear checks, the design shear force is checked to be less than the shear capacity found from clause 4.2.3. No allowance is made for the effect of boltholes when computing the shear capacity of the member.

For major and minor axis bending checks, the design bending moment is checked to be less than the moment capacity as found using clause 4.2.5. **Note that the moment capacity is conservatively computed on the basis of interaction with the design shear force**.

For the lateral torsion buckling check, the design bending moment about the major principle axis is checked to be less than the buckling resistance moment as computed using clause 4.3.6 and annex B.2**.**

For tension checks, the design axial tensile force is checked to be less than the tension capacity of the member as computed using clause 4.6 with reference to Annex I.2. The capacity of single angle, channel and tee section member is computed using clause 4.6.3.1 if the specified bolt holes indicate that the member is connected via only the flange or web as appropriate. **Clauses 4.6.3.2 and 4.6.3.3 are not considered.**

The compression section check is a supplemental check not explicitly covered by BS5950. It checks that the design axial compressive force is less than the compressive section capacity that is computed as the product of the gross area of the section and the design strength of the steel (i.e. $P_c=A_g p_y$*).*

For major and minor compression buckling checks, the design axial compressive force in each column segment is checked to be less than the compressive resistance of each column segment as computed using clause 4.7.5 with specific reference to Annex C.1 and Annex C.2. **Clauses 4.7.6 to 4.7.13 are NOT considered.**

For all combined action section checks, the design axial forces (F_t and F_c) is the maximum tensile and compressive axial forces in the member, and the design bending moments (M_x, and M_y) are the maximum bending moments in the member.

For the combined axial tension and bending check, the design bending and axial force are checked to determine if they satisfy clause 4.8.2.

For the combined axial compression and bending checks, the design bending and axial force are checked to determine if they satisfy clause 4.8.3.

The auxiliary combined action checks consider a combination of two actions and take the value of the action not considered as zero.

For combined biaxial checks, the design bending moments are checked to satisfy clause 4.9.

For the combined axial tension and major bending check, the design bending and axial force are checked to determine if they satisfy clause 4.8.2 taking the value of My as zero.

Similarly, the combined axial tension and minor bending check, the design bending and axial force are checked to determine if they satisfy clause 4.8.2 taking the value of Mx as zero.

For the combined axial compression and major bending checks, the design bending and axial force are checked to determine if they satisfy clause 4.8.3 taking the value of My as zero.

For the combined axial compression and minor bending checks, the design bending and axial force are checked to determine if they satisfy clause 4.8.3 taking the value of Mx as zero.

Chapter 7
AS/NZS4600

This release note explains the AS/NZS4600 design code in Multiframe Steel Codes. It provides a step-by-step description of how to modify the design properties used by the code.

- Setting Properties
- Bending
- Tension
- Compression
- Combined Actions
- Design Properties
- Steel Grade
- Code Checks
- References

Setting Properties - AS/NZS4600

Before doing the checks, it is necessary to enter basic design data such as effective length, grade of steel etc. This information can either be entered in the Frame window by selecting members and using the commands under the Design menu, or it can be entered in tabular form in the Data window. All of the windows and commands which are common to Multiframe work the same way in Multiframe Steel Codes. You have all the display options of Multiframe and facilities to help you select the required members using clipping, masking etc. In general you can not change the frame or its loading in Multiframe Steel Codes, the only change you can make is to change the section for a member. If you do change a section, you will need to re-analyse using the Analyse command.

Although most of the design variables are pre-set to the most commonly used values, you will probably want to enter the design information for at least some of the members in the frame that you wish to check. You set design variables by selecting the members you wish to change and then choosing the appropriate command from the Design menu.

There are a number of design variables which are used when doing checking to the code. A summary of all of the design variables is as follows;

Variable Name	Description	Default Value
Fy	Yield strength of the section's steel	250Mpa
Fu	Ultimate Tensile Strength of the section's steel	320Mpa
Kx	Effective length factor for buckling about the section's strong axis	1.0
Ky	Effective length factor for buckling about the section's weak axis	1.0
Lcx	Unbraced length for preventing column buckling about the section’s strong axis.	member's length
Lcy	Unbraced length for preventing column buckling about the section’s weak axis.	member's length

Lateral restraints	The lateral restraints acting on the member.	Each end of the member is fully restrained at both flanges.
ds	Length of stiffeners. Assume that all stiffeners has the same length regardless of web stiffeners or flange stiffeners	0.0 (ie no stiffeners)
s1	Edge distance between the first stiffener and the element edge. Assume that all stiffeners on a web or flange are symmetric to the centre line of the element.	0.0 (ie no stiffeners)
s2	The distance between the first and the second stiffener. Assume that all stiffeners on a web or flange are symmetric to the centre line of the element.	0.0 (ie less than 3 stiffeners)
No. of stiffeners	Number of stiffeners. This is either the total amount of stiffeners on web(s) or the total amount of stiffeners on flange(s). eg. for a C section with 8 stiffeners on flanges, so each flange has 8/2 = 4 stiffeners. However, for a back-to-back section with 8 stiffeners, each flange has 8/4 = 2 stiffeners.	0 (i.e. no stiffeners)
No. of Flange Holes	The number of holes in the flanges of the section.	0
Diameter of Flange Holes	Diameter of holes in the flanges of the section.	0.0
Total Height of Flange Holes	Total height of any bolt holes in the flanges of the section. This value may be input directly or computed automatically when the number and diameter of flange holes are specified.	0.0
No. of Web Holes	The number of holes in the webs of the section.	0
Diameter of Web Holes	Diameter of holes in the webs of the section.	0.0
Total Height of Web Holes	Total height of any bolt holes in the webs of the section. This value may be input directly or computed automatically when the number and diameter of flange holes are specified.	0.0
kt	Correction factor for the distribution of forces.	1.0
Max Depth	The maximum depth of section which may be chosen when using the Design command	Depth of the initial section
Min Depth	The minimum depth of section which may be chosen when using the Design command	depth of the initial section
Max Width	The maximum width of section which may be chosen when using the Design command	width of the initial section
Min Width	The minimum width of section which may be chosen when using the Design command	width of the initial section

C_s	Moment coefficient. +1.0 for moment causing compression on shear centre side of the centroid while -1.0 for moment causing tension on shear centre side of the centroid.	1.0
C_b	Coefficient depending on moment distribution in the laterally unbraced segment.	1.0
C_{mx}	Coefficient for unequal end moment.	1.0
C_{my}	Coefficient for unequal end moment.	1.0
R	Purlins' reduction factor. For channel- and Z-purlins in which the tension flange is attached to sheeting, the member bending capacity subjected to lateral buckling is calculated with clause 3.3.3.4.	1.0

It is not necessary to enter all of the above information for all members. Usually you will want to check some members for bending, others for compression and so on. The items under the Design menu help you enter just the required information depending on what type of check you are doing.

Bending - AS/NZS4600

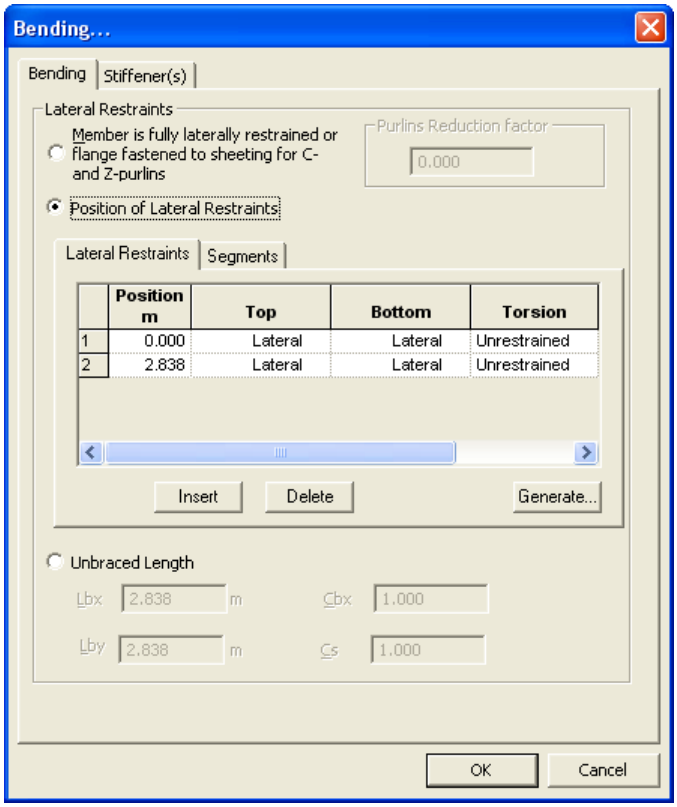
When performing a bending check, you may need to specify the location and type of lateral restraints acting on the member. It is also necessary to enter the stiffener's information.

To determine the moment member capacity of a member, it is necessary to know the spacing of any lateral restraints (if any) along the member. The restraints could be provided by purlins, girts or other structural elements which are not modelled in Multiframe. Multiframe Steel Codes uses this information to determine the length of segments used in the design calculations. The lateral restraints acting at a particular section on a member are dependent upon which flange is the critical flange. For a member/segment restrained at both ends the critical flange is the flange under compression. For a cantilever or a segment with an unrestrained end, the critical flange is the tension flange. For each restraint on the member, the user must specify the type of restraint. As this depends upon which flange is the critical flange, the user must specify the type of lateral restraint that would be present at a section if

i) the top flange were the critical flange, and
ii) the bottom flange was the critical flange.

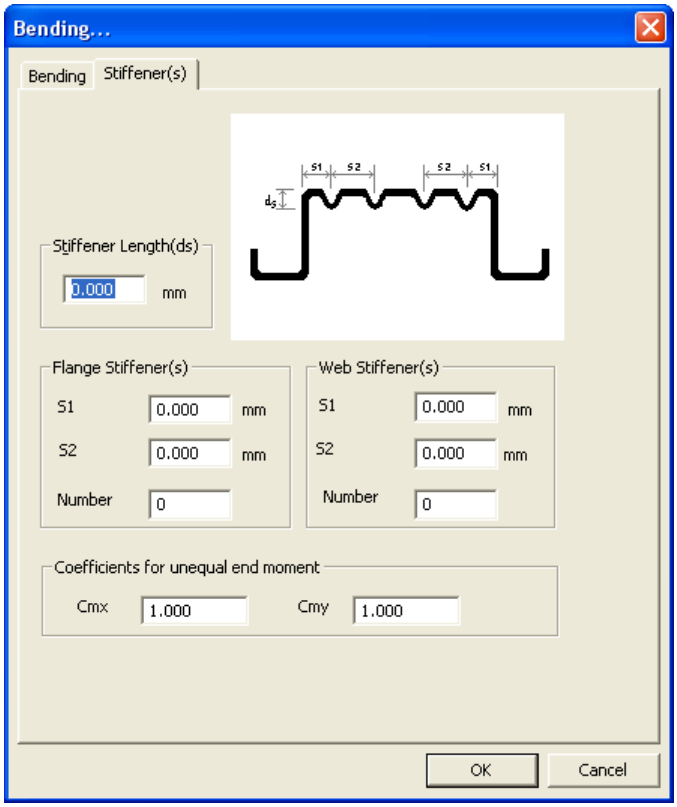
To set the properties for bending

- **Select the required members in the Frame window.**
- **Choose Bending from the Design menu.**



- **Click the type of lateral restraints.**
- **Enter the position and type of lateral restraints for both top and bottom flange.**

If there are transverse stiffeners on the web or flange, click the stiffener tab and see the following window.



- **Enter the length of stiffener**
- **Enter the number of stiffeners and spacing(s) etc.**
- **Enter coefficients for unequal end moment**
- **Click OK**

Lateral restraints must always be specified at the ends of the beam and so the minimum number of lateral restraints is two. If no restraint exists at the end of a member then it should be specified as unrestrained. The initial lateral restraints applied to the member are full restraints at each end for either of the flanges being the critical flange.

The different restraints acting on the member are entered into the grid using the following codes;

F Fully restrained
P Partially restrained
L Laterally Restrained
U Unrestrained
LR Lateral restraint with full restraint against rotation on plan
LP Lateral restraint with partial restraint against rotation on plan
C Continuous restraint

Fully or partially restrained sections may also be specified as lateral rotational restraints using;

FR Fully restrained + Rotationally restrained
PR Partial restrained + Rotationally restrained

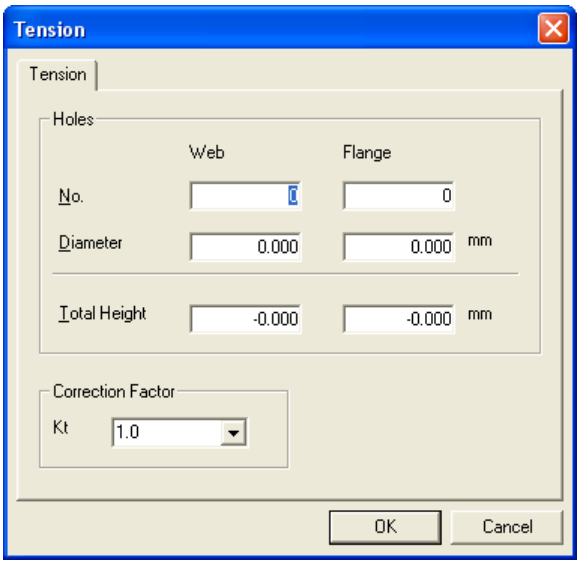
The initial position of the loads is at the shear centre. If there are no transverse stiffeners, leave the stiffener spacing set to zero.

Tension - AS/NZS4600

When checking or designing a member for tension, you need to specify the correction factor for the distribution of forces at the ends of the member. If the members contain significant areas of bolt holes which need to be taken into account when determining the cross-sectional area of the section, you will need to enter the amount of cross-sectional area to be deducted to allow for these holes.

To enter the properties for tension

- **Select the required members in the Frame window**
- **Choose Tension… from the Design menu**



- **Type in the number and diameter of holes in the webs and flanges (and the total height of holes will be computed automatically) or**
- **Type the total height of holes in the webs and flanges directly**
- **Choose a value for the correction factor (kt) if required**
- **Click OK**

The total height of holes in the webs or flanges is used to compute the cross sectional area of holes in the section. This is used compute the net area of the section and also for computing the effective section modulus. The initial value for the number and diameter of bolt holes is zero.

When checking or designing members for compression, it is necessary to specify the effective length and unbraced length of the member.

Compression - AS/NZS4600

To determine the critical buckling load for a member, it is necessary to enter an effective length to indicate the type of restraint on the ends of the member. The effective length is given by an effective length factor multiplied by the length of the member. The effective length may be different for buckling in the major and minor axis directions. The effective lengths are given by

$$L_{ex} = K_x \cdot L_{cx} \text{ and } L_{ey} = K_y \cdot L_{cy},$$

where

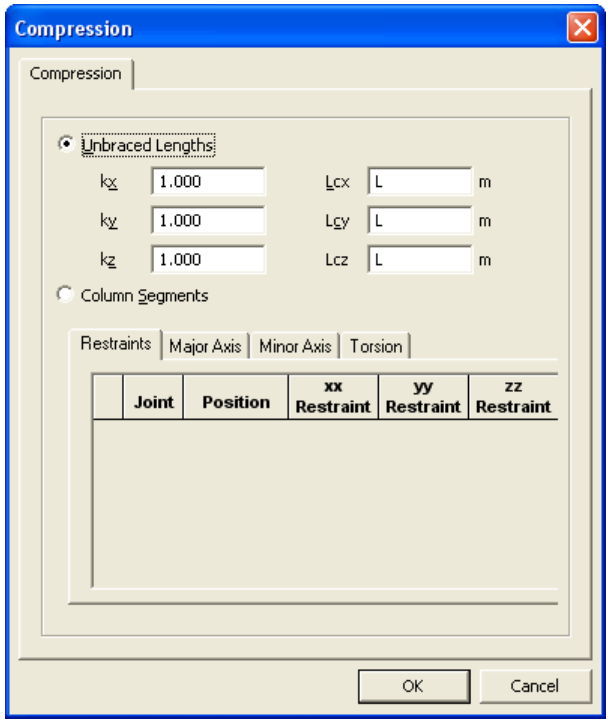
L_{cx} and L_{cy} are the lengths of the member in x and y direction respectively, K_x and K_y are the two effective length factors for the major and minor axes respectively. The initial values of K_x and K_y are 1.0.

Unbraced Length - AS/NZS4600

To determine the critical buckling condition of a member, it is also necessary to know the spacing of any bracing (if any) along the member. This bracing could be provided by purlins, girts or other structural elements which are not modelled in Multiframe. Some bracing may only restrain lateral deflection in one direction, therefore it is necessary to enter unbraced lengths for both axes of the section, L_{cx} corresponding to the spacing of restraints preventing compression buckling about the x-x axis and L_{cy} corresponding to the spacing of restraints preventing compression buckling about the y-y axis.

To set the properties for compression

- **Select the required members in the Frame window**
- **Choose Compression… from the Design menu**



- **Click on the icons for the end conditions in each direction or**
- **Type in values for K_x and K_y**

- **Type in values for L_{cx} and L_{cy}**
- **Click OK**

If you choose a standard end condition, the recommended K_x and K_y values will be automatically entered for you.

The initial values of L_{cx} and L_{cy} are the length of the member.

Combined Actions - AS/NZS4600

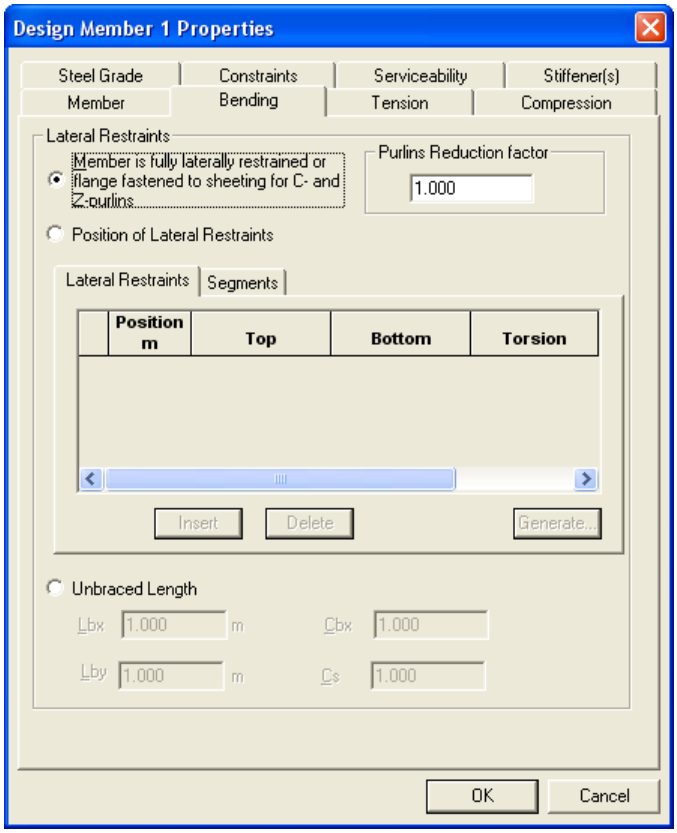
No information is required when checking or designing members for combined actions using AS/NZS4600.

Design Properties - AS/NZS4600

Sometimes you may wish to set all of the design properties for a member or group of members at once. This may be quicker than setting each of the design values in turn using the commands above.

To set all of the design variables

- **Select the required members in the Frame window**
- **Choose Design Details from the Design menu**



- **Click each tab and enter the design values**
- **Click OK**

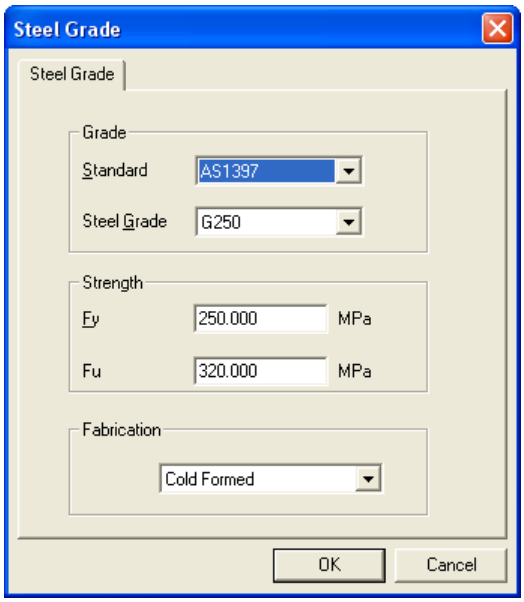
As a shortcut, you can examine and change the design details for a single member by double clicking on it in the Frame window.

Steel Grade - AS/NZS4600

To determine the allowable stresses for a member, it is necessary to know the grade of steel to be used for the section. This grade determines the yield strength (F_y) and ultimate tensile strength (F_u) of the material of the section.

To set the Steel Grade

- **Select the required members in the Frame window**
- **Choose Steel Grade from the Design menu**



In this dialog you can either

- **Choose a standard and steel grade from the drop down menu,**

or

- **Type in values for F_y and F_u.**

Finally

- **Choose the method of fabrication to indicate the state of residual stress in the section.**
- **Click OK.**

If you choose a standard grade of steel, the F_y and F_u values will be automatically entered for you.

The initial value for the steel grade for all members is AS1397 grade 250.

Code Checks - AS/NZS4600

When carrying out code checks to AS/NZS4600, Multiframe Steel Codes uses the following clauses of to check your structure. No other checks are performed unless they are specifically listed below.

AS/NZS 4600: "Australian/New Zealand Standard AS/NZS4600-2005: Cold-formed Steel Structures", Standards Australia, 30 December, 2005.

Clauses used are 3.1~3.5.

Design Checking Procedure

The design checking procedure is as follows;

The design actions are calculated through the first order analyses and a second order analysis should be used for sway frames.

For major and minor bending section checks, the design bending moment is checked to be less than the nominal section moment design capacity as found using clause 3.3.2.

For bending member checks, the design bending moment about the major principle axis is checked to be less than the nominal member moment design capacity as found using clause 3.3.3. For some section shapes, the bending of distortional buckling check may not be included: clause 3.3.3.3.

For major and minor shear checks, the design shear force is checked to be less than the nominal shear capacity found from section 3.3.4.

For tension checks, the design axial tension force is checked to be less than the nominal section design capacity in tension as computed using clause 3.2.

For compression section checks, the design axial compressive force is checked to be less than the nominal section design capacity in compression as computed using clause 3.4.1.

For major and minor compression member checks, the design axial compressive force is checked to be less than the nominal member design capacity in compression as computed using clause 3.4.2~3.4.5.

For all combined action section checks, the design axial force (N*) is the maximum axial force in the member, and the design bending moments (Mx*, and My*) are the maximum bending moments in the member.

For major and minor combined section checks, the design bending moment is checked to be less than the nominal section moment design capacity reduced by axial force (compression or tension) as computed using clause 3.5.1.

References - AS/NZS4600

You may find the following books useful to refer to if you need information on the methods used to check members in Multiframe Steel Codes.

- **Australian/New Zealand Standard AS/NZS 4600:2005, Cold-formed Steel Structures, Australian Institute of Steel Construction, Sydney, 1998, 3rd Edition**

- **Design of Cold-formed Steel Structures (to Australian/New Zealand Standard AS/NZS 4600:1996), J. Handcock, Australian Institute of Steel Construction, Sydney, 1998, 3rd Edition**
- **Design of Cold-formed Steel Members, J. Rhodes, Department of Mechanical Engineering, University of Strathclyde, Glasgow, UK, 1991**
- **Multiframe Steel Codess Handbook, B.Gorenc, R. Tinyou and A. Syam, UNSW Press, Sydney, 1996, 6th Edition**
- **The Behaviour and Design of Steel Structures, N S Trahair and M A Bradford, Chapman and Hall, London, 1988**

Chapter 8
AISI

This section explains the AISI design code in Multiframe Steel Codes. It provides a step-by-step description of how to modify the design properties used by the code.

- Setting Properties
- Bending
- Tension
- Compression
- Combined Actions
- Design Properties
- Steel Grade
- Code Checks References

Setting Properties - AISI

Before performing design checks, it is necessary to enter basic design data such as effective length, grade of steel etc. This information can either be entered in the Frame window, by selecting members and using the commands under the Design menu, or it can be entered in tabular form in the Design Details tab of the Data window.

Although most of the design variables are pre-set to the most commonly used values, you will probably want to enter the design information for at least some of the members in the frame that you wish to check. You set design variables by selecting the members you wish to change and then choosing the appropriate command from the Design menu.

There are a number of design variables which are used when doing checking to the code. A summary of all of the design variables is as follows;

Variable Name	Description	Default Value
Fy	Yield strength of the section's steel	250Mpa
Fu	Ultimate Tensile Strength of the section's steel	320Mpa
Kx	Effective length factor for buckling about the section's strong axis	1.0
Ky	Effective length factor for buckling about the section's weak axis	1.0
Lcx	Unbraced length for preventing column buckling about the section’s strong axis.	member's length
Lcy	Unbraced length for preventing column buckling about the section’s weak axis.	member's length
Lateral restraints	The lateral restraints acting on the member.	Each end of the member is fully restrained at both flanges.
ds	Length of stiffeners. Assume that all stiffeners have the same length regardless of whether they are web stiffeners or flange stiffeners	0.0 (ie no stiffeners)

s1	Edge distance between the first stiffener and the element edge. Assume that all stiffeners on a web or flange are symmetric to the centre line of the element.	0.0 (ie no stiffeners)
s2	The distance between the first and the second stiffener. Assume that all stiffeners on a web or flange are symmetric to the centre line of the element.	0.0 (ie less than 3 stiffeners)
No. of stiffeners	Number of stiffeners. This is either the total number of stiffeners on the web(s) or the total number of stiffeners on the flange(s). eg. for a C section with 8 stiffeners on flanges, so each flange has 8/2 = 4 stiffeners. However, for a back-to-back C section with 8 stiffeners, each flange has 8/4 = 2 stiffeners.	0 (i.e. no stiffeners)
No. of Flange Holes	The number of holes in the flanges of the section.	0
Diameter of Flange Holes	Diameter of holes in the flanges of the section.	0.0
Total Height of Flange Holes	Total height of any bolt holes in the flanges of the section. This value may be input directly or computed automatically when the number and diameter of flange holes are specified.	0.0
No. of Web Holes	The number of holes in the webs of the section.	0
Diameter of Web Holes	Diameter of holes in the webs of the section.	0.0
Total Height of Web Holes	Total height of any bolt holes in the webs of the section. This value may be input directly or computed automatically when the number and diameter of flange holes are specified.	0.0
kt	Correction factor for the distribution of forces.	1.0
Max Depth	The maximum depth of section which may be chosen when using the Design command	Depth of the initial section
Min Depth	The minimum depth of section which may be chosen when using the Design command	depth of the initial section
Max Width	The maximum width of section which may be chosen when using the Design command	width of the initial section
Min Width	The minimum width of section which may be chosen when using the Design command	width of the initial section
C_s	Moment coefficient. +1.0 for moment causing compression on shear centre side of the centroid while -1.0 for moment causing tension on shear centre side of the centroid.	1.0
C_b	Coefficient depending on moment distribution in the laterally unbraced segment.	1.0
C_{mx}	Coefficient for unequal end moment.	1.0

C_{my}	Coefficient for unequal end moment.	1.0
R	Purlins' reduction factor. For channel- and Z-purlins in which the tension flange is attached to sheeting, the member bending capacity subjected to lateral buckling is calculated with clause 3.3.3.4.	1.0

It is not necessary to enter all of the above information for all members. Usually you will want to check some members for bending, others for compression and so on. The items under the Design menu help you enter just the required information depending on what type of check you are doing.

Bending - AISI

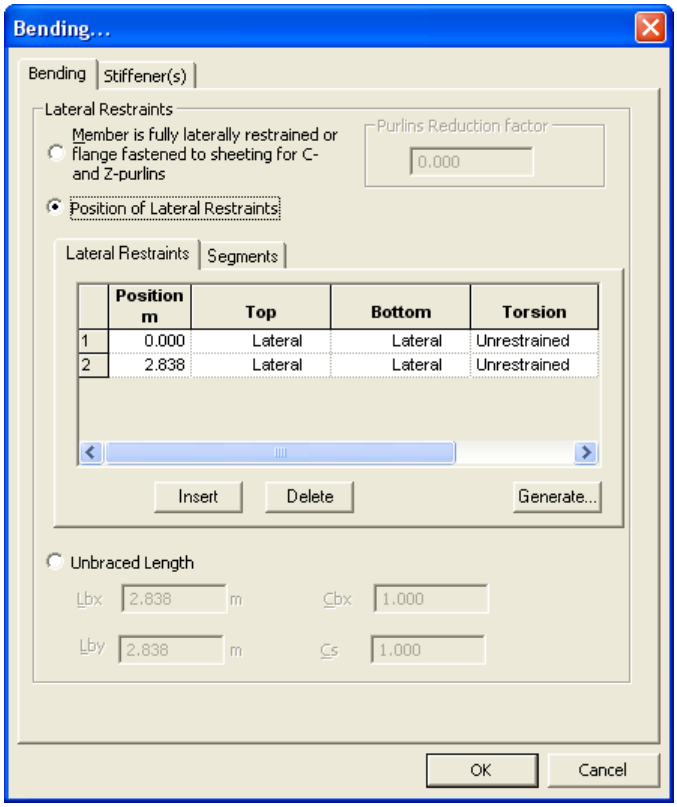
When performing a bending check, you may need to specify the location and type of lateral restraints acting on the member. It is also necessary to enter the stiffener's information.

To determine the moment member capacity of a member, it is necessary to know the spacing of any lateral restraints (if any) along the member. The restraints could be provided by purlins, girts or other structural elements which are not modelled in Multiframe. Multiframe Steel Codes uses this information to determine the length of segments used in the design calculations. The lateral restraints acting at a particular section on a member are dependent upon which flange is the critical flange. For a member/segment restrained at both ends the critical flange is the flange under compression. For a cantilever or a segment with an unrestrained end, the critical flange is the tension flange. For each restraint on the member, the user must specify the type of restraint. As this depends upon which flange is the critical flange, the user must specify the type of lateral restraint that would be present at a section if

i) the top flange were the critical flange, and
ii) the bottom flange was the critical flange.

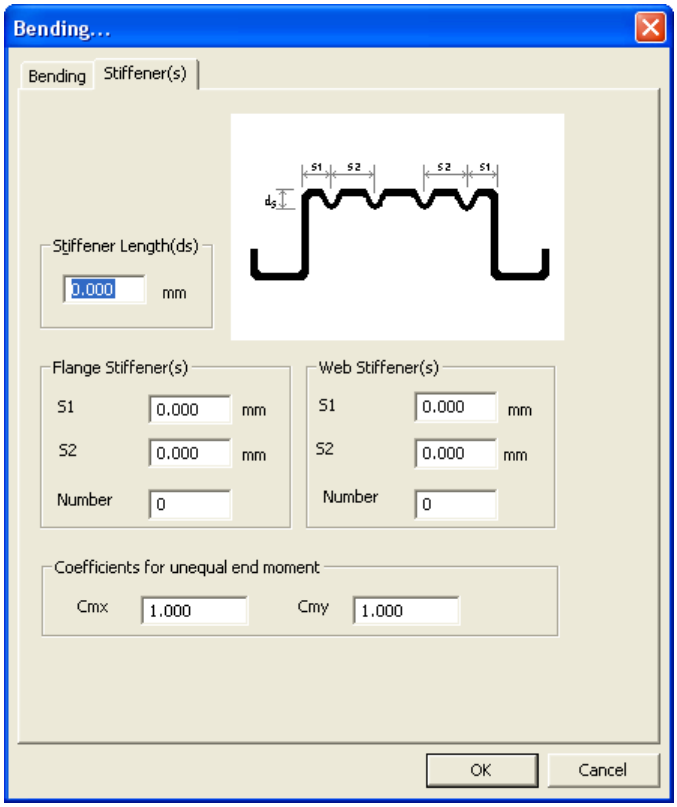
To set the properties for bending

- **Select the required members in the Frame window.**
- **Choose Bending from the Design menu.**



- **Click the type of lateral restraints.**
- **Enter the position and type of lateral restraints for both top and bottom flange.**

If there are transverse stiffeners on the web or flange, click the stiffener tab and see the following window.



- **Enter the length of stiffener**
- **Enter the number of stiffeners and spacing(s) etc.**
- **Enter coefficients for unequal end moment**
- **Click OK**

Lateral restraints must always be specified at the ends of the beam and so the minimum number of lateral restraints is two. If no restraint exists at the end of a member then it should be specified as unrestrained. The initial lateral restraints applied to the member are full restraints at each end for either of the flanges being the critical flange.

The different restraints acting on the member are entered into the grid using the following codes;

F Fully restrained
P Partially restrained
L Laterally Restrained
U Unrestrained
LR Lateral restraint with full restraint against rotation on plan
LP Lateral restraint with partial restraint against rotation on plan
C Continuous restraint

Fully or partially restrained sections may also be specified as lateral rotational restraints using;

FR Fully restrained + Rotationally restrained
PR Partial restrained + Rotationally restrained

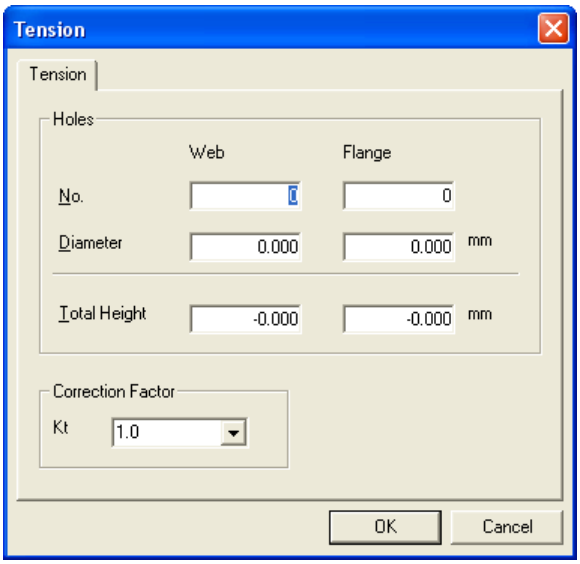
The initial position of the loads is at the shear centre. If there are no transverse stiffeners, leave the stiffener spacing set to zero.

Tension - AISI

When checking or designing a member for tension, you need to specify the correction factor for the distribution of forces at the ends of the member. If the members contain significant areas of bolt holes which need to be taken into account when determining the cross-sectional area of the section, you will need to enter the amount of cross-sectional area to be deducted to allow for these holes.

To enter the properties for tension

- **Select the required members in the Frame window**
- **Choose Tension… from the Design menu**



- **Type in the number and diameter of holes in the webs and flanges (and the total height of holes will be computed automatically) or**
- **Type the total height of holes in the webs and flanges directly**
- **Choose a value for the correction factor (kt) if required**
- **Click OK**

The total height of holes in the webs or flanges is used to compute the cross sectional area of holes in the section. This is used compute the net area of the section and also for computing the effective section modulus. The initial value for the number and diameter of bolt holes is zero.

When checking or designing members for compression, it is necessary to specify the effective length and unbraced length of the member.

Compression - AISI

To determine the critical buckling load for a member, it is necessary to enter an effective length to indicate the type of restraint on the ends of the member. The effective length is given by an effective length factor multiplied by the length of the member. The effective length may be different for buckling in the major and minor axis directions. The effective lengths are given by

$$L_{ex} = K_x \cdot L_{cx} \text{ and } L_{ey} = K_y \cdot L_{cy},$$

where

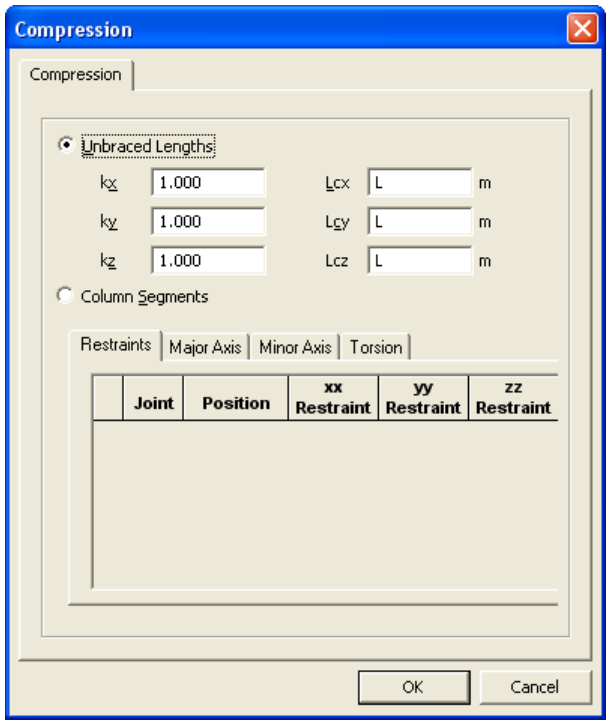
L_{cx} and L_{cy} are the lengths of the member in x and y direction respectively, K_x and K_y are the two effective length factors for the major and minor axes respectively. The initial values of K_x and K_y are 1.0.

Unbraced Length - AISI

To determine the critical buckling condition of a member, it is also necessary to know the spacing of any bracing (if any) along the member. This bracing could be provided by purlins, girts or other structural elements which are not modelled in Multiframe. Some bracing may only restrain lateral deflection in one direction, therefore it is necessary to enter unbraced lengths for both axes of the section, L_{cx} corresponding to the spacing of restraints preventing compression buckling about the x-x axis and L_{cy} corresponding to the spacing of restraints preventing compression buckling about the y-y axis.

To set the properties for compression

- **Select the required members in the Frame window**
- **Choose Compression… from the Design menu**



- **Click on the icons for the end conditions in each direction or**
- **Type in values for K_x and K_y**

- **Type in values for L_{cx} and L_{cy}**
- **Click OK**

If you choose a standard end condition, the recommended K_x and K_y values will be automatically entered for you.

The initial values of L_{cx} and L_{cy} are the length of the member.

Combined Actions - AISI

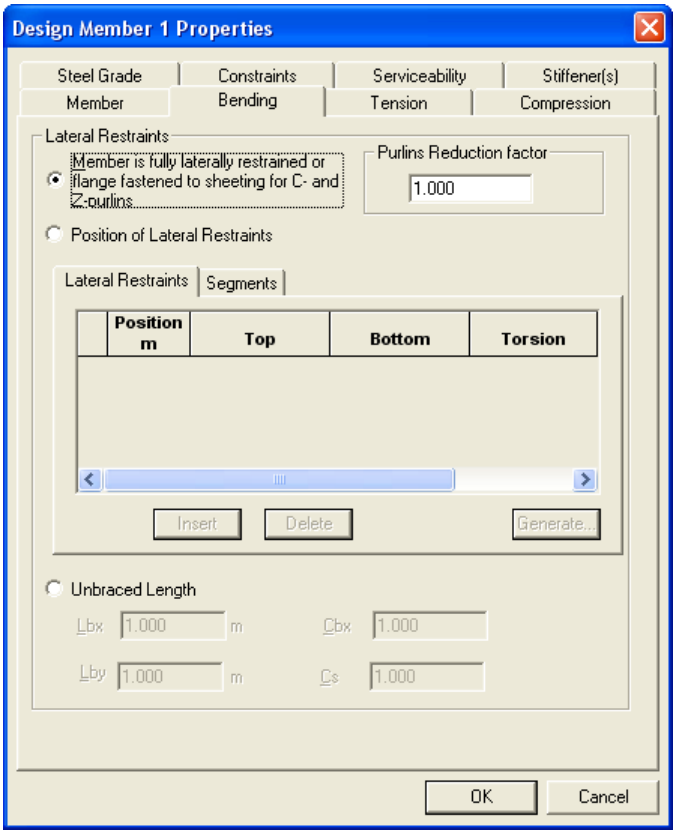
No information is required when checking or designing members for combined actions using AISI.

Design Properties - AISI

Sometimes you may wish to set all of the design properties for a member or group of members at once. This may be quicker than setting each of the design values in turn using the commands above.

To set all of the design variables

- **Select the required members in the Frame window**
- **Choose Design Details from the Design menu**



- **Click each tab and enter the design values**
- **Click OK**

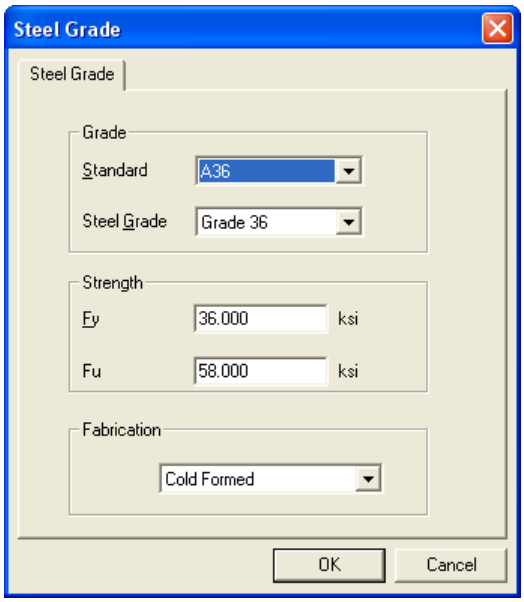
As a shortcut, you can examine and change the design details for a single member by double clicking on it in the Frame window.

Steel Grade - AISI

To determine the allowable stresses for a member, it is necessary to know the grade of steel to be used for the section. This grade determines the yield strength (F_y) and ultimate tensile strength (F_u) of the material of the section.

To set the Steel Grade

- **Select the required members in the Frame window**
- **Choose Steel Grade from the Design menu**



In this dialog you can either

- **Choose a standard and steel grade from the drop down menu,**

or

- **Type in values for F_y and F_u.**

Finally

- **Choose the method of fabrication to indicate the state of residual stress in the section.**
- **Click OK.**

If you choose a standard grade of steel, the F_y and F_u values will be automatically entered for you.

The initial value for the steel grade for all members is A36 grade 36.

Code Checks - AISI

When carrying out code checks to AISI, Multiframe Steel Codes uses the following clauses of to check your structure. No other checks are performed unless they are specifically listed below.

AISI: "North American Specification for the Design of Cold-formed Steel Structural Members ", AISI Standards, 2001 Edition.

Clauses used are C2~C5.

Design Checking Procedure

The design checking procedure is as follows;

The design actions are calculated through the first order analyses and a second order analysis should be used for sway frames.

For major and minor bending section checks, the design bending moment is checked to be less than the nominal section moment design capacity as found using clause C3.

For bending member checks, the design bending moment about the major principle axis is checked to be less than the nominal member moment design capacity as found using clause C3.1.

For major and minor shear checks, the design shear force is checked to be less than the nominal shear capacity found from section C3.2.

For tension checks, the design axial tension force is checked to be less than the nominal section design capacity in tension as computed using clause C2.

For compression section checks, the design axial compressive force is checked to be less than the nominal section design capacity in compression as computed using clause C4.

For major and minor compression member checks, the design axial compressive force is checked to be less than the nominal member design capacity in compression as computed using clause C4.1~C4.6.

For all combined action section checks, the design axial force (P*) is the maximum axial force in the member, and the design bending moments (Mx*, and My*) are the maximum bending moments in the member.

For major and minor combined section checks, the design bending moment is checked to be less than the nominal section moment design capacity reduced by axial force (compression or tension) as computed using clause C5.

References - AISI

You may find the following books useful to refer to if you need information on the methods used to check members in Multiframe Steel Codes.

- **Cold-formed Steel Design, Wei-Wen Yu, John Wiley & Sons, Inc., New York, 2000, 3rd Edition**
- **Design of Cold-formed Steel Structures to the AISI Specification, Gregory J. Handcock, Thomas M. Murray and Duane S. Ellifritt, Marcel Dekker, Inc., New York, 2001**

- **Design of Cold-formed Steel Members, J. Rhodes, Department of Mechanical Engineering, University of Strathclyde, Glasgow, UK, 1991**
- **Multiframe Steel Codess Handbook, B.Gorenc, R. Tinyou and A. Syam, UNSW Press, Sydney, 1996, 6th Edition**
- **The Behaviour and Design of Steel Structures, N S Trahair and M A Bradford, Chapman and Hall, London, 1988**

Chapter 9
AISC 2005/2010

This chapter describes the implementation of the AISC "Specification for Structural Steel Buildings" within Multiframe Steel Codes. It provides a step-by-step description of how to modify the design properties used by the code.

The AISC 2005 is a single, unified structural design code replacing the previously separate Load and Resistance Factor Design (LRFD) and Allowable Stress Design (ASD) codes. The only differences between the application of these two codes are in the use of design capacity factors and for the calculation of C_b for the use in calculation of resistance to biaxial bending combined with a axial force.

The updated AISC 2010 code is also implemented.

In the LRFD version of the code the allowable strength is given by

$$P_c = \phi P_n$$

Where ϕ is the resistance factor (always less than 1.0)
P_n the design strength

For ASD calculations the allowable strength is given by

$$P_c = P_n / \Omega$$

Where Ω is the safety factor (always greater than 1.0)
P_n the design strength

Values for resistance factors (ϕ – LRFD) and safety factors (Ω – ASD) for the various strength checks are set in Multiframe Steel Codes.

- Notation
- Design Checks
- Bending
- Tension
- Compression
- Combined Actions
- Serviceability
- Default Design Properties
- Code Clauses Checked

Notation – AISC 2005/2010

The notation used in Multiframe Steel Codes generally follows that used in the AISC design code. Use has been made of subscripts to clarify the axis of the member to which a quantity refers. For example, the nominal flexural strengths about the X and Y axes are denoted M_{nx} and M_{ny} respectively.

The geometric axes of a member are denoted as the X and Y axes where X represented the horizontal axis of the member and Y the vertical axis of the member. For design to AISC 2005, it is assumed that the X axis is the major axis and Y is the minor axis.

Design Checks - AISC 2005/2010

The types of checks are grouped into the categories; Bending, Tension, Compression, Combined and Serviceability. The user may specify which of these checks are performed when a member is designed or checked using Multiframe Steel Codes.

Bending - AISC 2005/2010

The design of a member for bending is divided into four design checks. These check the flexural and shear capacity of the member about the major and minor axes. Each of these checks may consider one or more limit states depending upon the section and the actions within the member.

When performing a bending check it is necessary to specify how lateral buckling of the member is resisted. Restraint could be provided by other members, purlins, girts or by other structural elements that are not modelled in Multiframe such as concrete slabs. Multiframe Steel Codes provides three methods of specifying how a member is restrained against lateral buckling. The user may specify

That the member is fully restrained against lateral buckling in which case no lateral buckling checks will be performed.
The location and type of lateral restraints applied to the member in which case Multiframe Steel Codes will appropriately divide the member into a number of spans and consider the capacity of each of these spans in determining the capacity of the member.

Alternatively the laterally unbraced length (L_b) can be specified.

You may need to specify a number of properties relating to the location and type of lateral restraints and the stiffener spacing along the member

Lateral Restraints - AISC 2005/2010

If the spacing of lateral restraints along the member is specified, Multiframe Steel Codes uses this information to break the member up into a number of spans in order to determine lateral torsion buckling capacity of each span. In Multiframe Steel Codes, these spans are known as segments.

Each lateral restraint specified by the user is assumed to provide bracing against lateral displacement of the *critical flange* and/or prevent twist of the cross section. At any cross section, the *critical flange* is the flange that, in the absence of any restraint at that cross section, would deflect the furthest during buckling of the member. In most members the *critical flange* will be the compression flange. However for a cantilevered member, the *critical flange* is the tension flange.

For each restraint located along a member, the user must specify the type of restraint. As this depends upon which flange is the critical flange, which is not know *a priori*, the user must specify the type of lateral restraint that would be present at a section if

- **The top flange was the critical flange, and**
- **The bottom flange was the critical flange.**

In AISC 2005 no distinction is made between different types of lateral restraints. However, to be compatible with other design codes, Multiframe Steel Codes allows for lateral restraints at a cross section to be classified as follows

- **Full Restraint –supports the cross section against lateral displacement of the critical flange and prevents twist of the cross section.**
- **Partial Restraint – provides support against lateral displacement of the section at a point other than the critical flange and prevents twist of the cross section.**
- **Lateral Restraint – resists lateral displacement of the critical flange only.**

For the purpose of design in AISC 2005/2010, each of these restraint types is consider adequate to provide lateral support to the cross section at which they are applied.

Lateral restraints must always be specified at the ends of the beam and so the minimum number of lateral restraints is two. If no restraint exists at the end of a member then it should be specified as unrestrained in which case the member would be regarded as a cantilever. The initial lateral restraints applied to the member are full restraints at each end for either of the flanges being the critical flange.

The location and type of lateral restraints can be displayed in the Frame and Plot windows. The display of lateral restraints can be turned on or off via the Symbols Dialog which contains options for displaying and labelling lateral restraints. The restraints are drawn as a short line in the plane of the major axis of the member. These lines extend each side of the member for a distance that is roughly the scale of a purlin or girt. Lateral restraints are also displayed in the rendered view of the frame in which they are draw to extend from each flange by approximately the size of a purlin. The restraints may be labelled using a one or two letters to indicate the type of restraint (e.g. F - fixed, P – partial, L - lateral).

Note that lateral restraints at the end of a member are draw slightly offset from the node so that restraints at the ends of connected members may be more readily distinguished.

Unbraced Length (L_b) - AISC 2005/2010

Instead of specifying the position of lateral restraints it may be preferable to directly set the laterally unbraced length of the member (L_b).

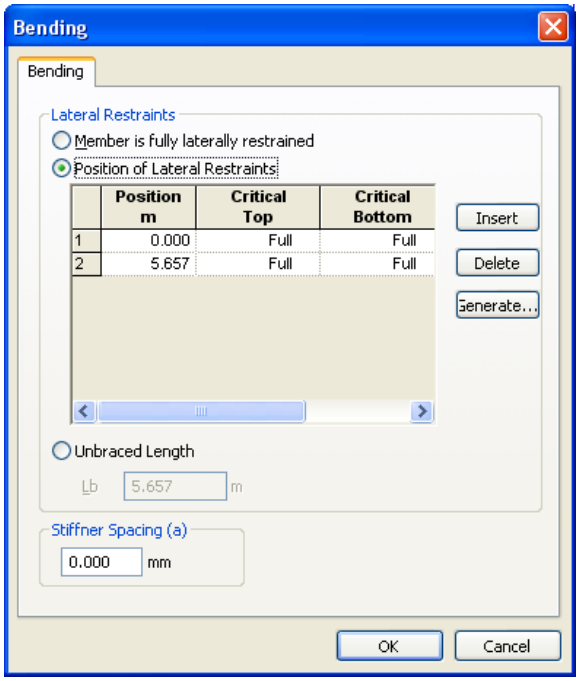
Web Stiffener Spacing - AISC 2005/2010

When checking or designing a member for bending, you may need to specify the spacing of any stiffeners along the web of the member. This affects the member's susceptibility to buckling due to bending. If there are no transverse stiffeners, you should leave the stiffener spacing set to zero.

Bending Dialog AISC 2005/2010

To set the properties for bending

- **Select the required members in the Frame window**
- **Choose Bending from the Design menu**



- **Select the "Member is fully laterally restrained" option, or**
- **Select the "Position of Lateral Restraints" option, and then**

To add new restraint to the member

- **Position the cursor with the table and click the Insert button to add a lateral restraint to the member.**
- **Select the position of each restraint**
- **Select the type of each lateral restraint from the combo provided in each cell.**

or

- **Click the Generate button to automatically generate a number of restraints.**

To delete a restraint from the member

- **Position the cursor within the table on the lateral restraint to be deleted and click the Delete button.**

To define the unbraced length

- **Select the "Unbraced Length" option, and then**
- **Enter the unbraced length (L_b)**

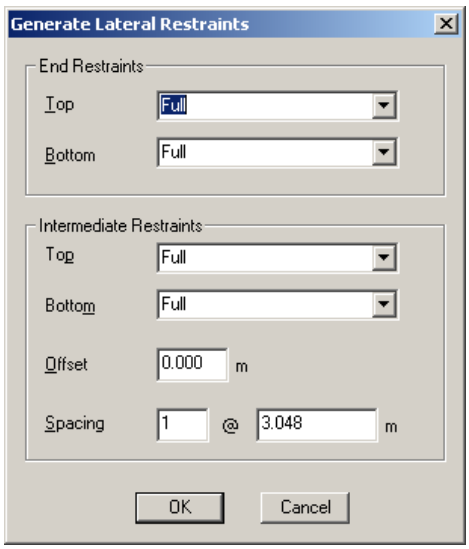
To define the stiffener spacing

- **If there are transverse stiffeners on the web, type in values for the stiffener spacing (a)**
- **Click OK**

Generate Lateral Restraints Dialog - AISC 2005/2010

When the user selects to generate the lateral restraints from the Bending dialog, the Generate Lateral Restraints dialog is displayed. This dialog enables the user to generate lateral restraints are a specified spacing along the member.

- **From the Bending dialog, click the Generate… button**



- **Select the type of restraints to be used at the ends of the member**
- **Select the type of restraints to be used at intermediate points within the member**
- **Enter the offset length at which the first intermediate restraint will be positioned. Leave this field as zero if no offset is same as the spacing**
- **Enter the number and size of spacing for the intermediate restraints.**
- **Click OK**

All lateral restraint applied to the member will now be regenerated and will replace all existing restraints.

Tension - AISC 2005/2010

The capacity of a member to resist tensile forces is implemented as a single design check. A number of modification factors may be entered to change the section properties used for checking tension. This includes the area of holes in the flange or web of the member and a shear lag factor to account for the distribution of forces at the ends of a member.

In addition to checking the tensile capacity of the member, a design constraint will be applied to the member enforcing the slenderness of the member to be less than 300.

Bolt Holes - AISC 2005/2010

When checking or designing a member for tension, you need to specify any reduction in area due to boltholes or other openings within the section. The net area of the section is the gross area minus the combined area of boltholes in the flange and web. In computing net area the diameter of a bolthole shall be taken as 1/16 in. (2mm) greater than the nominal dimension of the hole.

For a chain of holes extending in a diagonal or zigzag line, the net width of the section is obtained by deducting the sum of the diameters of all holes in the chain and adding for each gage space in the chain the quantity $s^2/4g$ where s is the longitudinal centre to centre spacing (pitch) of any two consecutive holes and g is the transverse centre to centre spacing (gage) between fastener gage lines.

The reduction in area can be specified by setting the number, diameter, pitch and gage of holes in the web or flanges of the member.

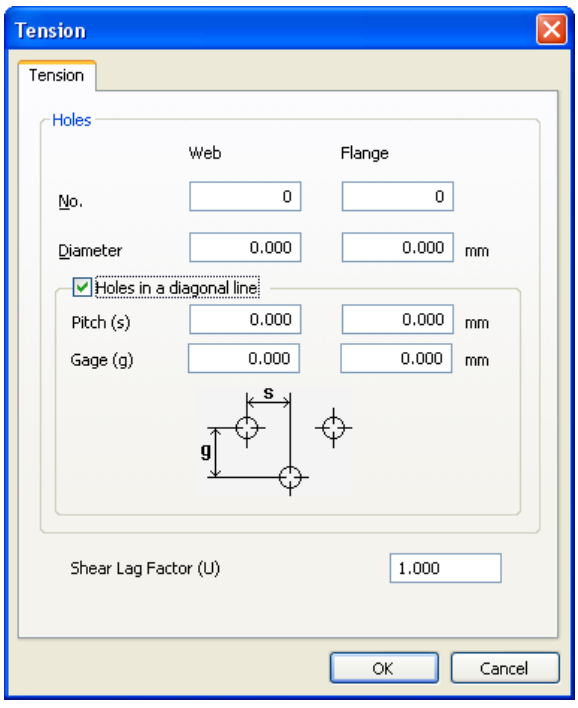
Shear Lag Factor - AISC 2005/2010

When checking or designing a member for tension using AISC 2005, you need to specify the reduction coefficient for the distribution of forces at the ends of the member. This coefficient is used to factor the net area in order to compute the effective area. The Shear Lag Factor, U, has a default value of 1.0

Tension Dialog - AISC 2005/2010

To enter the properties for tension

- **Select the required members in the Frame window**
- **Choose Tension… from the Design menu**



- **Type in the number and diameter of holes in the webs and flanges**

- **If the holes extend in a diagonal or zigzag line check the Holes in Diagonal Line box and enter the pitch and gage of holes in the webs and flanges**
- **Enter a value for the Shear Lag Factor (U)**
- **Click OK**

Compression - AISC 2005/2010

To determine the critical buckling load for a member, it is necessary to enter an effective length to indicate the type of restraint on the ends of the member. The effective length is given by an effective length factor multiplied by the length of the member. The effective length may be different for buckling in the major and minor axis directions. The effective lengths are given by

$L_{ex} = K_x L_{cx}$, $L_{ey} = K_y L_{cy}$ and $L_{ez} = K_z L_{cz}$

where

L_{cx} and L_{cy} are the lengths of the member in x and y direction respectively,
K_x and K_y are the two effective length factors for the major and minor axes respectively.
L_{cz} and K_z are the effective length and effective length factors to resist torsional buckling.
The initial values of K_x, K_y and K_z are 1.0.

Unbraced Length - AISC 2005/2010

To determine the critical buckling condition of a member, it is also necessary to know the spacing of any bracing (if any) along the member. This bracing could be provided by purlins, girts or other structural elements which are not modelled in Multiframe. Some bracing may only restrain lateral deflection in one direction, therefore it is necessary to enter unbraced lengths for both axes of the section, L_{cx} corresponding to the spacing of restraints preventing compression buckling about the x-x axis and L_{cy} corresponding to the spacing of restraints preventing compression buckling about the y-y axis. It is also possible to enter L_{cz} and K_z, used in the calculation of torsional buckling resistance, at this point.

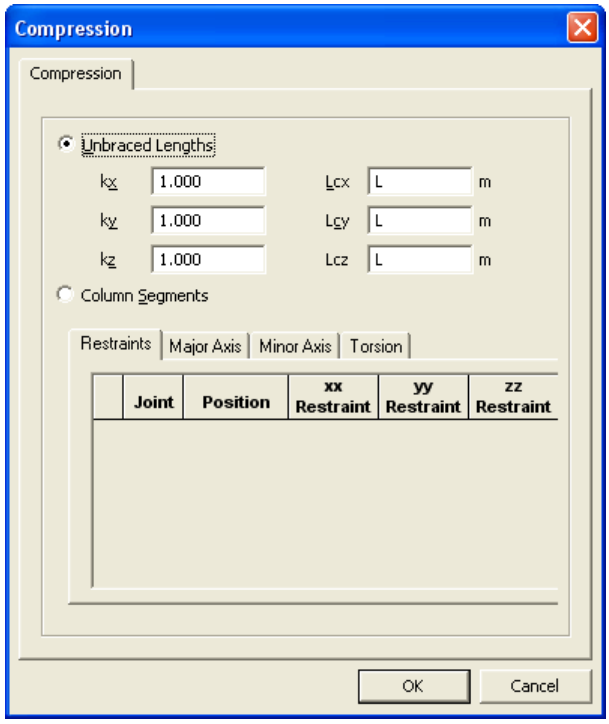
Compression Dialog – AISC 2005/2010

To set the properties for compression

- **Select the required members in the Frame window**
- **Choose Compression… from the Design menu**

If the unbraced lengths of the member are to be specified directly then

- **Select the Unbrace Length radio button.**



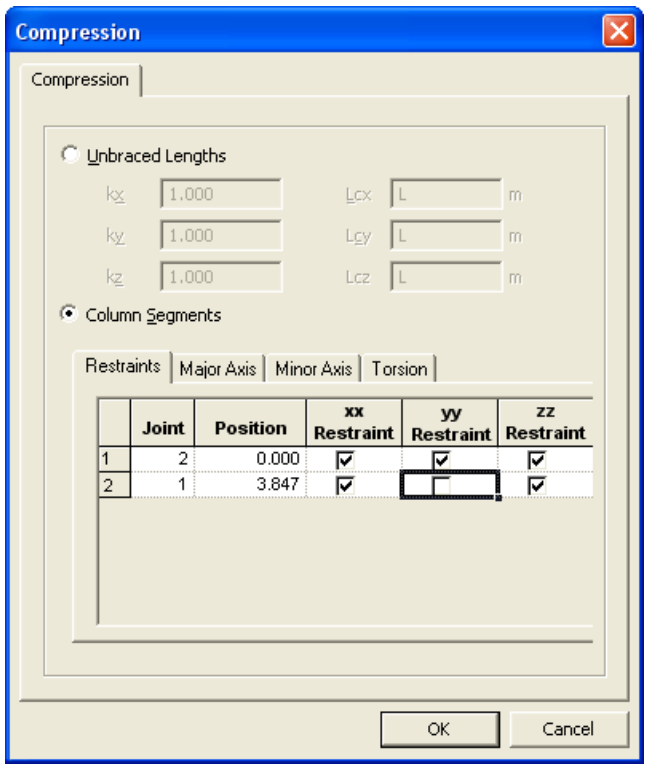
- **Type in values for Kx, Ky and Kz**
- **Type in values for Lcx, Lcy and Lcz**
- **Click OK**

The initial values of L_{cx}, L_{cy} and L_{cz} are the length of the member. The default values of K_x, K_y and K_z are 1.0.

Otherwise if the design for compression is to be performed using column segments.

- **Select the Column Segments radio button.**

The tabbed control in the dialog will become active. The first page in this table lists the location of joints along the members and indicates if they provide restraint against column bucking about either axis of the member.

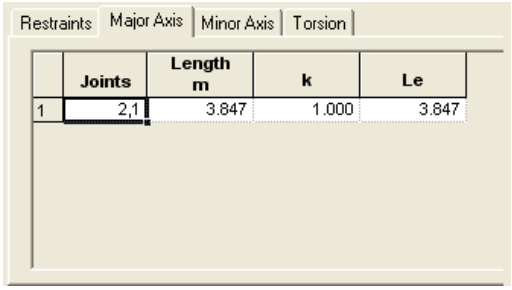


- **Enter the restraints associated with each node.**

The restraint information is used to build a list of column segments that span between the specified restraints.

- **Click on the Major Axis tab.**

This displays a table of column segments that will be used for the design of the member for compression when considering buckling about the major axis.



- **Enter the effective length factor (K) for each segment.**
- **Click on the Minor Axis tab and enter the effective length factors for the minor axis column segments.**
- **Click on the Torsion tab and enter the effective length factors for the calculation of torsional buckling resistance.**
- **Click OK**

Combined Actions – AISC 2005/2010

The design of a member for combined actions is divided into three design checks. The user can select to check the member for torsion or biaxial bending in conjunction with either a tensile or compressive axial force. The user is not required to provide any additional design properties for the combined actions checks as it uses results already derived from the tension, compression and bending checks.

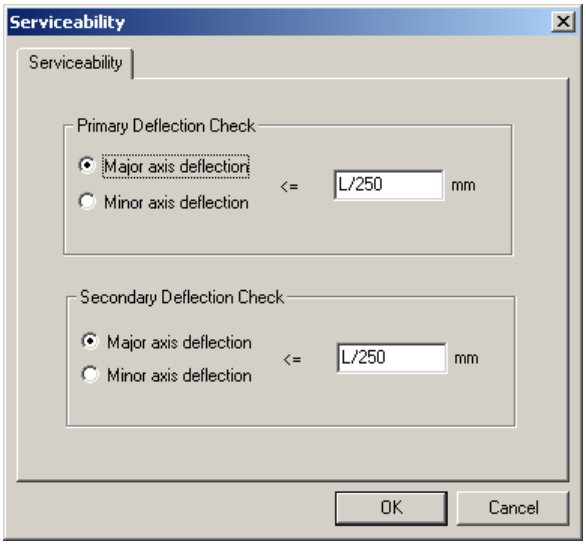
Serviceability - AISC 2005/2010

Multiframe Steel Codes provides two design checks for the serviceability of a member. These design checks are used to check that the deflection of a member about either the major or minor axes does not exceed a specified deflection limit.

Serviceability Dialog – AISC 2005/2010

To set the design properties of a member for serviceability

- **Select the required members in the Frame window**
- **Choose Serviceability … from the Design menu**



- **For each deflection check, select the axis about which the deflection will be checked.**
- **Type in values for the deflection limits.**
- **Click OK**

Default Design Properties – AISC 2005/2010

There are a number of design variables, which are used when doing checking to the code. A summary of all of the design variables is as follows;

Variable	Description	Default
F_y	Yield strength of the section's steel	250Mpa
F_u	Ultimate Tensile Strength of the section's steel	410Mpa

K_x	Effective length factor for buckling about the section's strong axis	1.0
K_y	Effective length factor for buckling about the section's weak axis	1.0
K_z	Effective length factor for torsional buckling.	1.0
L_{cx}	Unbraced length for bracing preventing buckling about the section's strong axis	Member's length
L_{cy}	Unbraced length for bracing preventing buckling about the section's weak axis	Member's length
L_{cy}	Unrestrained length for bracing preventing torsional buckling	Member's length
Lateral restraints	The lateral restraints acting on the member.	Each end of the member is fully restrained at both flanges.
L_b	Unrestrained length of member for lateral torsional buckling.	Member's length
a	Spacing of web stiffeners. This is the spacing of any stiffeners along the web of a beam	0.0 (i.e. no stiffeners)
No. of Flange Holes	The number of holes in the flanges of the section.	0
Diameter of Flange Holes	Diameter of holes in the flanges of the section.	0.0
Pitch of Flange Holes	Longitudinal spacing of staggered holes in the flanges of the section	0.0
Gage of Flange Holes	Transverse spacing of staggered holes in the flanges of the section	0.0
No. of Web Holes	The number of holes in the webs of the section.	0
Diameter of Web Holes	Diameter of holes in the webs of the section.	0.0
Pitch of Web Holes	Longitudinal spacing of staggered holes in the webs of the section	0.0
Gage of Web Holes	Transverse spacing of staggered holes in the webs of the section	0.0
U	Shear Lag Factor for the distribution of forces.	1.0
Fabrication	The method by which the section was manufactured. This describes the residual stresses in the section.	Hot Rolled

It is not necessary to enter all of the above information for all members. Usually you will want to check some members for bending, others for compression and so on. The items under the Design menu help you enter just the required information depending on what type of check you are doing.

Code Clauses Checked – AISC 2005/2010

When carrying out code checks, Multiframe Steel Codes uses the following clauses of the applicable codes to check your structure. No other checks are performed unless they are specifically listed below.

Checks are not carried out on composite members or tapered members. Checks using actions computed using plastic analysis are not considered.

"Specification for Structural Steel Buildings", American Institute of Steel Construction, March 9, 2005.

The design checking procedure is as follows:

The net area of the section is computed by subtracting the area of holes in the section. The effective area is then calculated as the net area (A_n) times the Shear Lag Factor (U).

If the member is been checked for tension of compression, the slenderness of the section is checked. For angle members, the slenderness about either of the geometric axes is determined using the minimum radius of gyration of the section.

For each serviceability load case:

The maximum local displacement of the member is compared to the deflection limits specified deflection limits.

For each load case representing a strength limit state,

The design actions, or required strengths, of the member are determined as the maximum moment, shears and axial forces within the member.

For first order analyses, the design bending moments are amplified using the moment amplification factors. Only moment amplification of braced frames is considered which corresponds to the situation in which no moments result from the lateral translation of the frame. Amplification factors for sway frames are not considered and a second order analysis should be used for sway frames requiring moment amplification.

The plate elements of the section will be classified as Compact/Non-Compact/Slender as per the requirements of Clause B4 and Table B4-1. If the moments in the member are less than one ten thousandth of the yield moments the section is considered to be in pure compression and will be classified accordingly. If an element of the section is found to be slender, the stiffness reduction factors Q, Q_a and Q_s will be determined as set out in Clause E7.

For Tension checks, the capacity of the member is determined in accordance with Chapter D.

For Compression checks, the capacity of the member is firstly computed for the limit states of flexural buckling about the major and minor axis is accordance with clause E3. The capacity of the member for the limit state of flexural torsional-buckling is then computed using clauses E4. The compressive capacity of the member is regarded as being the minimum capacity determined for these three limit states.

For Flexure checks the provisions of Chapter F are adhered to. Major and minor flexure checks are performed separately. The capacity of a member for the limit states of Yielding, Lateral-Torsional Buckling, Flange Local Buckling, Tension Flange Yielding, Flange Local Buckling and Web Local Buckling is computed. Not all limit states are applicable to every cross-section. These are detailed in Table F1.1. In addition flange local bucking will only be considered for sections with non-compact flanges. Similarly, web local buckling will only be considered for sections with non-compact webs.

The design for Shear is carried out in accordance with Chapter G. Major and minor shear checks are performed separately. Specified stiffener spacings are accounted for.

The combined cases of Torsion, Biaxial Bending and Axial loads are detailed in Chapter H. The resistance of a section to resistance torsional loading is calculated separately in accordance with Clause H3. Biaxial Bending and Axial loading capacity is a combination of the respective capacity checks carried out in previous chapters in accordance with Clause H1. For Doubly and Single Symmetric Members in Flexure and Tension the increase in the value of C_b varies between the LRFD and ASD versions of the code. This is detailed in Clause H1.2.

Chapter 10
Eurocode 3

This chapter describes the implementation of the EN 1-1-1993 “Specification for Structural Steel Buildings” within Multiframe Steel Codes. It provides a step-by-step description of how to modify the design properties used by the code.

- Notation
- Design Checks
- Bending
- Tension
- Compression
- Serviceability
- National Annex
- Default Design Properties
- Code Clauses Checked

Notation – Eurocode 3

The notation used in Multiframe Steel Codes generally follows that used in the EC3 design code. Use has been made of subscripts to clarify the axis of the member to which a quantity refers. For example, the nominal flexural strengths about the Y and Z axes are denoted $M_{y,Rd}$ and $M_{z,Rd}$ respectively.

The geometric axes of a member are denoted as the Y and Z axes where Y represented the horizontal axis of the member and Z the vertical axis of the member. For design to Eurocode 3, it is assumed that the Y axis is the major axis and Z is the minor axis.

Design Checks - Eurocode 3

The types of checks are grouped into the categories: Tension, Compression, Bending Torsion and Buckling. The user may specify which of these checks are performed when a member is designed or checked using Multiframe Steel Codes.

Bending - Eurocode 3

The design of a member for bending is divided into eight design checks. These check the flexural, shear and combined flexural-shear capacity of the member about the major and minor axes and the combined biaxial bending and axial force and the combined biaxial bending, shear and axial force. Each of these checks may consider one or more limit states depending upon the section and the actions within the member.

When performing a bending check it is necessary to specify how lateral buckling of the member is resisted. Restraint could be provided by other members, purlins, girts or by other structural elements that are not modelled in Multiframe such as concrete slabs. Multiframe Steel Codes provides three methods of specifying how a member is restrained against lateral buckling. The user may specify that the member is fully restrained against lateral buckling in which case no lateral buckling checks will be performed.

The location and type of lateral restraints applied to the member in which case Multiframe Steel Codes will appropriately divide the member into a number of spans and consider the capacity of each of these spans in determining the capacity of the member.

Alternatively the laterally unbraced length (L_b) can be specified.

You may need to specify a number of properties relating to the location and type of lateral restraints and the stiffener spacing along the member

Lateral Restraints - Eurocode 3

If the spacing of lateral restraints along the member is specified, Multiframe Steel Codes uses this information to break the member up into a number of spans in order to determine lateral torsion buckling capacity of each span. In Multiframe Steel Codes, these spans are known as segments.

Each lateral restraint specified by the user is assumed to provide bracing against lateral displacement of the *critical flange* and/or prevent twist of the cross section. At any cross section, the *critical flange* is the flange that, in the absence of any restraint at that cross section, would deflect the furthest during buckling of the member. In most members the *critical flange* will be the compression flange. However for a cantilevered member, the *critical flange* is the tension flange.

For each restraint located along a member, the user must specify the type of restraint. As this depends upon which flange is the critical flange, which is not know *a priori*, the user must specify the type of lateral restraint that would be present at a section if

- **The top flange was the critical flange, and**
- **The bottom flange was the critical flange.**

In Eurocode 3 no distinction is made between different types of lateral restraints. However, to be compatible with other design codes, Multiframe Steel Codes allows for lateral restraints at a cross section to be classified as follows

- **Full Restraint –supports the cross section against lateral displacement of the critical flange and prevents twist of the cross section.**
- **Partial Restraint – provides support against lateral displacement of the section at a point other than the critical flange and prevents twist of the cross section.**
- **Lateral Restraint – resists lateral displacement of the critical flange only.**

For the purpose of design in Eurocode 3, each of these restraint types is consider adequate to provide lateral support to the cross section at which they are applied.

Lateral restraints must always be specified at the ends of the beam and so the minimum number of lateral restraints is two. If no restraint exists at the end of a member then it should be specified as unrestrained in which case the member would be regarded as a cantilever. The initial lateral restraints applied to the member are full restraints at each end for either of the flanges being the critical flange.

The location and type of lateral restraints can be displayed in the Frame and Plot windows. The display of lateral restraints can be turned on or off via the Symbols Dialog which contains options for displaying and labelling lateral restraints. The restraints are drawn as a short line in the plane of the major axis of the member. These lines extend each side of the member for a distance that is roughly the scale of a purlin or girt. Lateral restraints are also displayed in the rendered view of the frame in which they are draw to extend from each flange by approximately the size of a purlin. The restraints may be labelled using a one or two letters to indicate the type of restraint (e.g. F - fixed, P – partial, L - lateral).

Note that lateral restraints at the end of a member are draw slightly offset from the node so that restraints at the ends of connected members may be more readily distinguished.

Unbraced Length (L_b) - Eurocode 3

Instead of specifying the position of lateral restraints it may be preferable to directly set the laterally unbraced length of the member (L_b).

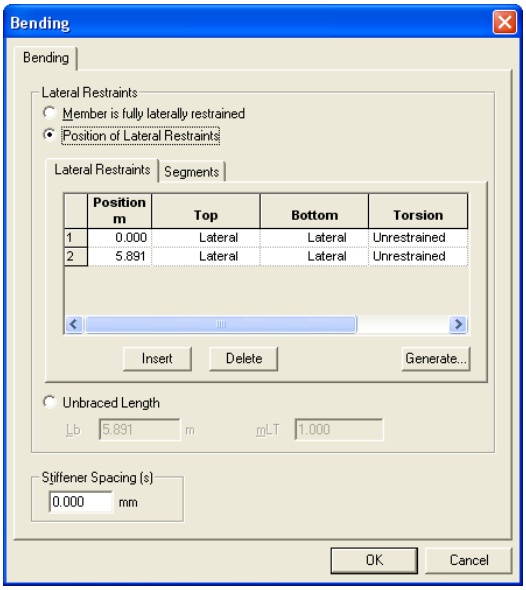
Web Stiffener Spacing - Eurocode 3

When checking or designing a member for bending, you may need to specify the spacing of any stiffeners along the web of the member. This affects the member's susceptibility to buckling due to bending. If there are no transverse stiffeners, you should leave the stiffener spacing set to zero.

Bending Dialog Eurocode 3

To set the properties for bending

- **Select the required members in the Frame window**
- **Choose Bending from the Design menu**



- **Select the "Member is fully laterally restrained" option, or**
- **Select the "Position of Lateral Restraints" option, and then**

To add new restraint to the member

- **Position the cursor with the table and click the Insert button to add a lateral restraint to the member.**
- **Select the position of each restraint**
- **Select the type of each lateral restraint from the combo provided in each cell.**

or

- **Click the Generate button to automatically generate a number of restraints.**

To delete a restraint from the member

- **Position the cursor within the table on the lateral restraint to be deleted and click the Delete button.**

To define the unbraced length

- **Select the "Unbraced Length" option, and then**
- **Enter the unbraced length (L_b)**

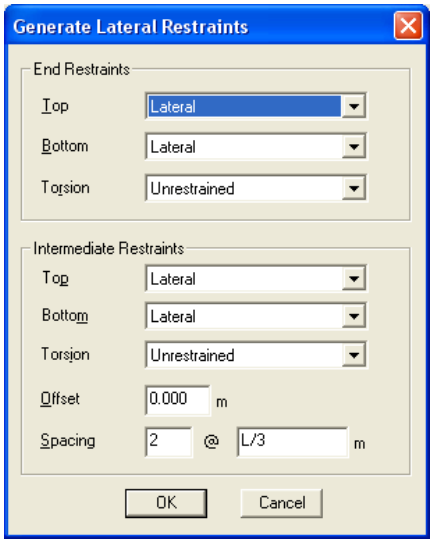
To define the stiffener spacing

- **If there are transverse stiffeners on the web, type in values for the stiffener spacing (a)**
- **Click OK**

Generate Lateral Restraints Dialog - Eurocode 3

When the user selects to generate the lateral restraints from the Bending dialog, the Generate Lateral Restraints dialog is displayed. This dialog enables the user to generate lateral restraints are a specified spacing along the member.

- **From the Bending dialog, click the Generate… button**



- **Select the type of restraints to be used at the ends of the member**
- **Select the type of restraints to be used at intermediate points within the member**
- **Enter the offset length at which the first intermediate restraint will be positioned. Leave this field as zero if no offset is same as the spacing**
- **Enter the number and size of spacing for the intermediate restraints.**
- **Click OK**

All lateral restraint applied to the member will now be regenerated and will replace all existing restraints.

Tension - Eurocode 3

The capacity of a member to resist tensile forces is implemented as a single design check. A number of modification factors may be entered to change the section properties used for checking tension. This includes the area of holes in the flange or web of the member and a shear lag factor to account for the distribution of forces at the ends of a member.

In addition to checking the tensile capacity of the member, a design constraint will be applied to the member enforcing the slenderness of the member to be less than 300.

Bolt Holes - Eurocode 3

When checking or designing a member for tension, you need to specify any reduction in area due to boltholes or other openings within the section. The net area of the section is the gross area minus the combined area of boltholes in the flange and web.

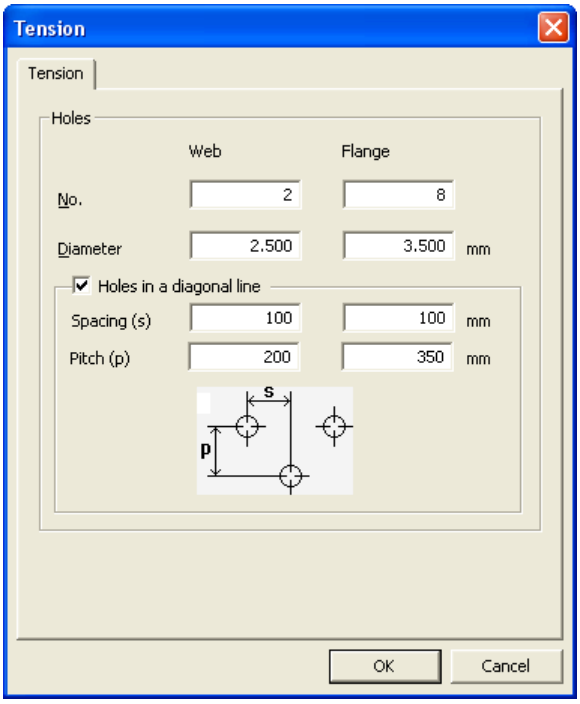
For a chain of holes extending in a diagonal or zigzag line, the net width of the section is obtained by deducting the sum of the diameters of all holes in the chain and adding for each gage space in the chain the quantity $s^2/4p$ where s is the longitudinal centre to centre *spacing* of any two consecutive holes and p is the transverse centre to centre *pitch* between fastener gage lines.

The reduction in area can be specified by setting the number, diameter, pitch and gage of holes in the web or flanges of the member.

Tension Dialog - Eurocode 3

To enter the properties for tension

- **Select the required members in the Frame window**
- **Choose Tension… from the Design menu**



- **Type in the number and diameter of holes in the webs and flanges**
- **If the holes extend in a diagonal or zigzag line check the Holes in Diagonal Line box and enter the Spacing and Pitch of holes in the webs and flanges**
- **Click OK**

Compression - Eurocode 3

To determine the critical buckling load for a member, it is necessary to enter an effective length to indicate the type of restraint on the ends of the member. The effective length is given by an effective length factor multiplied by the length of the member. The effective length may be different for buckling in the major and minor axis directions. The effective lengths are given by

$L_{ey} = K_y L_{cy}$ and $L_{ez} = K_z L_{cz}$

where

L_{cy} and L_{cz} are the lengths of the member in x and y direction respectively,
K_y and K_z are the two effective length factors for the major and minor axes respectively.
The initial values of K_y and K_z are 1.0.

Unbraced Length - Eurocode 3

To determine the critical buckling condition of a member, it is also necessary to know the spacing of any bracing (if any) along the member. This bracing could be provided by purlins, girts or other structural elements which are not modelled in Multiframe. Some bracing may only restrain lateral deflection in one direction, therefore it is necessary to enter unbraced lengths for both axes of the section, L_{cy} corresponding to the spacing of restraints preventing compression buckling about the y-y axis and L_{cz} corresponding to the spacing of restraints preventing compression buckling about the z-z axis.

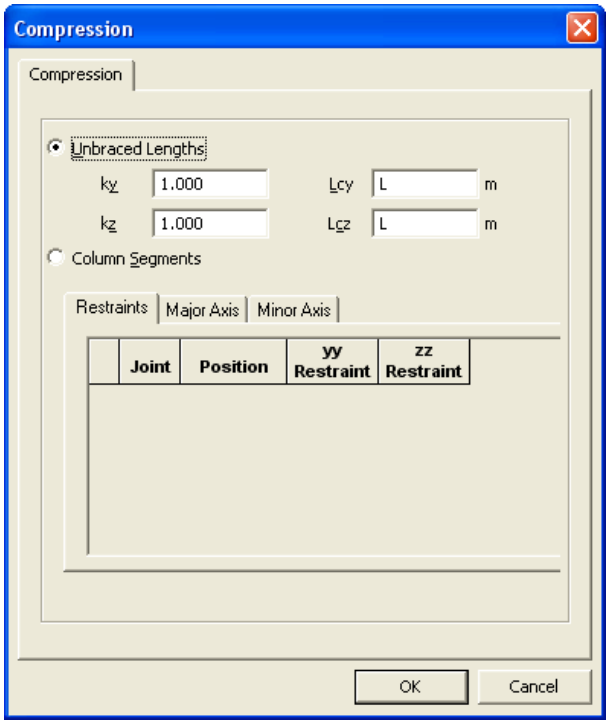
Compression Dialog – Eurocode 3

To set the properties for compression

- **Select the required members in the Frame window**
- **Choose Compression… from the Design menu**

If the unbraced lengths of the member are to be specified directly then

- **Select the Unbrace Length radio button.**



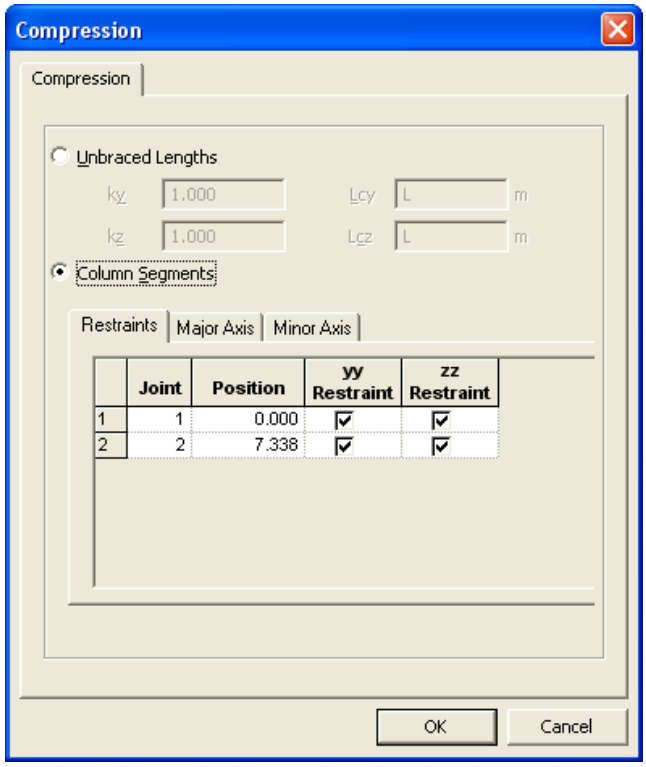
- **Type in values for ky and kz**
- **Type in values for Lcy and Lcz**
- **Click OK**

The initial values of L_{cy} and L_{cz} are the length of the member. The default values of k_y and k_z are 1.0.

Otherwise if the design for compression is to be performed using column segments.

- **Select the Column Segments radio button.**

The tabbed control in the dialog will become active. The first page in this table lists the location of joints along the members and indicates if they provide restraint against column bucking about either axis of the member.

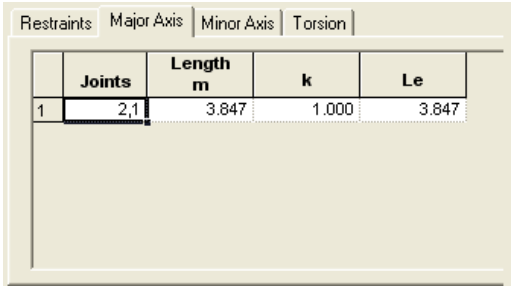


- **Enter the restraints associated with each node.**

The restraint information is used to build a list of column segments that span between the specified restraints.

- **Click on the Major Axis tab.**

This displays a table of column segments that will be used for the design of the member for compression when considering buckling about the major axis.



- **Enter the effective length factor (k) for each segment.**
- **Click on the Minor Axis tab and enter the effective length factors for the minor axis column segments.**
- **Click on the Torsion tab and enter the effective length factors for the calculation of torsional buckling resistance.**
- **Click OK**

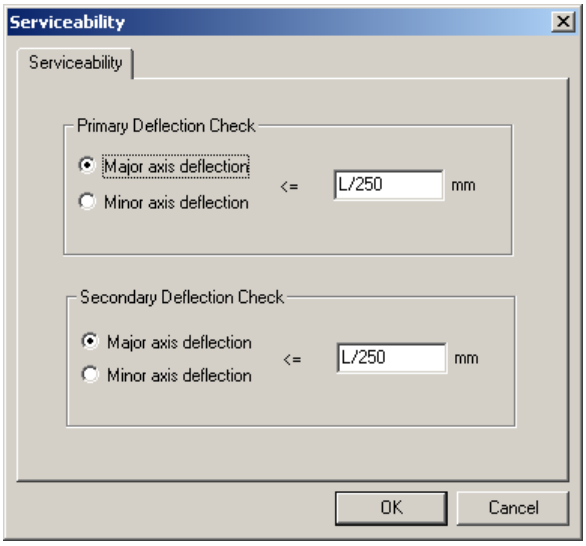
Serviceability - Eurocode 3

Multiframe Steel Codes provides two design checks for the serviceability of a member. These design checks are used to check that the deflection of a member about either the major or minor axes does not exceed a specified deflection limit.

Serviceability Dialog - Eurocode 3

To set the design properties of a member for serviceability

- **Select the required members in the Frame window**
- **Choose Serviceability … from the Design menu**



- **For each deflection check, select the axis about which the deflection will be checked.**
- **Type in values for the deflection limits.**
- **Click OK**

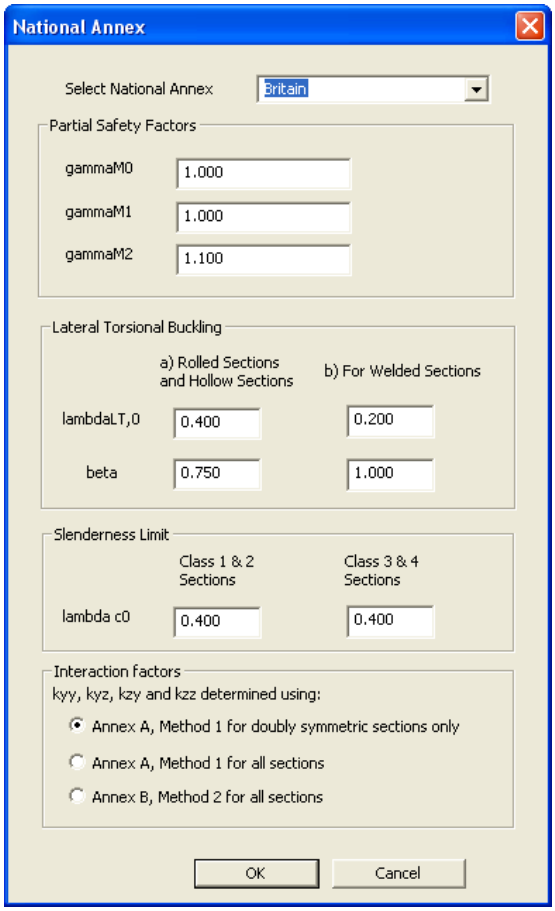
National Annex

Multiframe Steel Codes allows the choice of National Annex within Eurocode 3. Default values for nations supported can be used or properties can be manually entered.

National Annex Dialog – Eurocode 3

To set the National Annex properties for a model

- **Choose National Annex … from the Design menu (must have Eurocode 3 selected)**



- **From the Select National Annex drop down box choose the country you are working in. All other fields will be automatically populated. If your country is not available, choose Other**

- **If you have chosen Other or want to change any of the properties type in the desired values**

- **Click OK to save and use these selections**

Default Design Properties - Eurocode 3

There are a number of design variables, which are used when doing checking to the code. A summary of all of the design variables is as follows;

Variable	Description	Default
f_y	Yield strength of the section's steel	250Mpa
f_u	Ultimate Tensile Strength of the section's steel	410Mpa
k_y	Effective length factor for buckling about the section's strong axis	1.0
k_y	Effective length factor for buckling about the section's weak axis	1.0
L_{cy}	Unbraced length for bracing preventing buckling about the section's strong axis	Member's length
L_{cz}	Unbraced length for bracing preventing buckling about the section's weak axis	Member's length

Lateral restraints	The lateral restraints acting on the member.	Each end of the member is fully restrained at both flanges.
L_b	Unrestrained length of member for lateral torsional buckling.	Member's length
a	Spacing of web stiffeners. This is the spacing of any stiffeners along the web of a beam	0.0 (i.e. no stiffeners)
No. of Flange Holes	The number of holes in the flanges of the section.	0
Diameter of Flange Holes	Diameter of holes in the flanges of the section.	0.0
Staggered pitch of Flange Holes (s)	Spacing of fastener holes measured parallel to the member axis	0.0
Spacing of Flange Holes (p)	Transverse spacing of staggered holes in the flanges of the section	0.0
No. of Web Holes	The number of holes in the webs of the section.	0
Diameter of Web Holes	Diameter of holes in the webs of the section.	0.0
Staggered pitch of Web Holes (s)	Longitudinal spacing of staggered holes in the webs of the section	0.0
Spacing of Web Holes (p)	Transverse spacing of staggered holes in the webs of the section	0.0
Fabrication	The method by which the section was manufactured. This describes the residual stresses in the section.	Hot Rolled

It is not necessary to enter all of the above information for all members. Usually you will want to check some members for bending, others for compression and so on. The items under the Design menu help you enter just the required information depending on what type of check you are doing.

Code Clauses Checked – Eurocode 3

When carrying out code checks, Multiframe Steel Codes uses the following clauses of the applicable codes to check your structure. No other checks are performed unless they are specifically listed below.

EN 1993-1-1:2005 "Eurocode 3: Design of Steel Structures – Part 1-1: General rules and rules for buildings", May 2005

The design checking procedure is as follows:

The net area of the section is computed by subtracting the area of holes in the section.

For each serviceability load case:

The maximum local displacement of the member is compared to the deflection limits specified deflection limits.

For each load case representing a strength limit state,

The design actions, or required strengths, of the member are determined as the maximum moment, shears and axial forces within the member.

For first order analyses, the design bending moments are amplified using the moment amplification factors. Only moment amplification of braced frames is considered which corresponds to the situation in which no moments result from the lateral translation of the frame. Amplification factors for sway frames are not considered and a second order analysis should be used for sway frames requiring moment amplification.

The plate elements of the section will be classified as Class 1, 2, 3 or 4 as per the requirements of Section 5.5.2 and Table 5.2. In Class 4 cross sections effective widths are calculated to make the necessary allowances reductions in resistance to the effects of local buckling.

For Tension checks, the capacity of the member is determined in accordance with Chapter 6.2.3. The smaller of the values for design plastic resistance without considering fastener holes and the ultimate resistance including fastener holes is used.

For Compression checks, the capacity of the member is firstly computed using the area of the cross section for Class 1, 2 or 3 cross-sections. For Class 4 cross-sections the effective area is used.

For Bending checks the provisions of Chapter 6.2.5is adhered to. Major and minor flexure checks are performed separately. For Class 1 and 2 cross-sections are designed to their elastic limit. Class 3 and 3 cross-sections to their plastic limit, with Class 4 cross-sections using a reduced effective Plastic Modulus. At present no allowance is made for fastener holes.

The design for Shear is carried out in accordance with Chapter 6.2.6. Major and minor shear checks are performed separately.

Where shear force is present is allowed for in the combined Bending and Shear check as described in Chapter 6.2.8.

The combined cases of Bending and Axial force and Bending, Shear and Axial force are checked as described in Chapter 6.2.9. The Shear check is only included if present.

Torsion is detailed in Chapter 6.2.7. The torsional strength is a combination of the uniform torsional section resistance and the biomoment section resistance as per “The Behaviour and Design of Steel Structures to EC3” by Trahair et al.

The buckling cases of Compression Buckling, Lateral Torsional Buckling and Bending and Compression buckling are checked in accordance with Chapter 6.3. The Bending and Compression buckling interaction factors are calculated by either Method 1, detailed in EN 1993-1-1 Annex A or Method 2, detailed in EN 1993-1-1 Annex B. The decision as to which method to use depends on which National Annex is used or can be manually selected.

Chapter 11
User Code

User Codes - Concepts

At times, you may find you want to carry out design checks, which are different from those prescribed in the standard codes. To facilitate this, Multiframe Steel Codes has an additional code named User, which lets you enter design rules and check members according to these rules.

User Code – Procedures

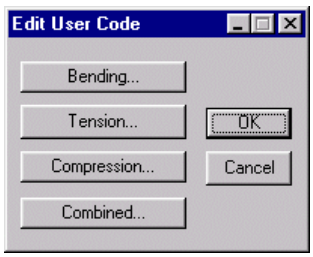
To activate the User code,

- **choose User from the Code menu.**

Now whenever you do any checking or designing, Multiframe Steel Codes will use the User code rules to determine a member's efficiency. You can view and edit the design rules in the User code by choosing the Edit User Code item from the Code menu. The rules in the User code are grouped into the four groups which appear in the Check and Design dialogs, that is Beams, Ties (or tension) Struts (or compression) and Beam-Columns (or combined).

To edit the User code

- **Choose Edit User Code… from the Code menu**



- **Click on the button of the part of the code you wish to change**

- **Type in new rules or modify the existing design rules**

The syntax of the design rules is the same as that of the Calculation sheet in Multiframe. This is very similar to the format used in most programming languages and spreadsheets. The following variables are available to help you construct your design rules. These variables are evaluated for each member as the member is checked.

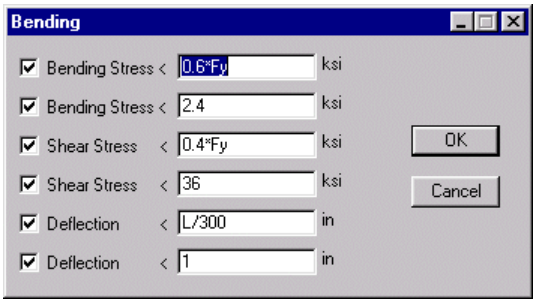
Variable	Value
L	Length of member*
Kx	Effective length factor in major plane
Ky	Effective length factor in minor plane
Lbx	Unbraced length for buckling about the major axis*
Lby	Unbraced length for buckling about the minor axis*
rx	radius of gyration about major axis*

ry	radius of gyration about minor axis*
E	Young's modulus of steel
ft	maximum tensile stress
fc	maximum compressive stress
fbx	maximum bending stress about major axis
fby	maximum bending stress about minor axis
fy	yield stress of the steel
fu	ultimate tensile strength of the steel
y	height of the highest end of the member above y=0*
a	web stiffener spacing*
Cb	bending coefficient
Cmx	major interaction coefficient
Cmy	minor interaction coefficient

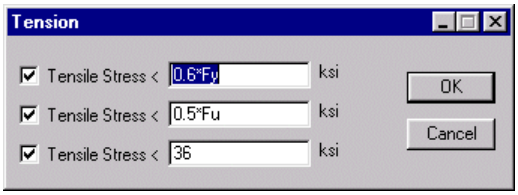
Note that all length variables (marked with an asterix * above) are given values in the same units as the units for deflection as specified in the Units dialog. This ensures that the dimensions of the resulting calculations will be consistent. All stresses and strengths have units as set for the Stresses option in the Units dialog.

The four different parts of the User code correspond to the four groups of checks available when using the Check and Design commands.

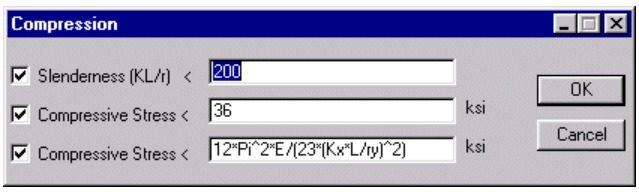
The bending checks can be used to check bending stresses, shear stresses and deflections. These formulas will be applied to both the major and minor axis beam calculations.



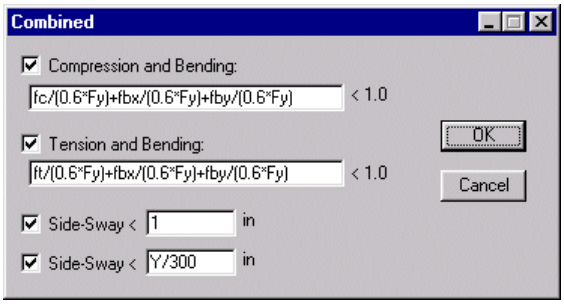
The tension checks will be used to evaluate the tensile stress on the member.



The compression checks will be used for the Slenderness and Compression check options when using the Check and Design commands.



The combined checks will be used for the Combined check options when using the Check and Design commands. The combined stress checks check the user formula against a combined stress ratio (CSR) of 1.0.



Only the calculations that have their check box checked will be used when you use the Check or Design commands.

Chapter 12
Multiframe Steel Codes Reference

This chapter summarises the extended functionality of windows and the extra menu commands that are available in Multiframe when Multiframe Steel Codes is enabled.

- Windows
- Menus

Windows

Multiframe Steel Codes operates within the standard Multiframe windows and adds a Report window. The following windows are available:

- Frame Window
- Data Window
- Load Window
- Result Window
- Plot Window
- Report Window

Frame Window

This window is used for specifying the sections and design properties of the members in a frame.

Data Window

This window is used for viewing the data describing the geometry and loading of the frame and for displaying and editing the design properties of the structure.

Load Window

This window is used for viewing the loading applied to the frame. One load case at a time may be viewed in this window. You can choose which load case is displayed by choosing the appropriate item from the bottom of the Case menu.

Result Window

This window is used for viewing the results of the analysis and design calculations carried out on the frame. The results for one load case at a time may be viewed in this window. You can choose which load case is displayed by choosing the appropriate item from the bottom of the Case menu. You can also view the Design Efficiency table in this window.

Plot Window

This window is used for viewing diagrams of the results of the analysis carried out on the frame. The results for one load case at a time may be viewed in this window. You can choose which load case is displayed by choosing the appropriate item from the bottom of the Case menu. You can also view a colour plot of design efficiency in this window.

Report Window

This window is used for viewing a summary report of the design checks carried out on the frame. You can turn on or off the option to create a summary report when you use the Check or Design commands.

Menus

When the Multiframe Steel Codes module is active some extra menu items are displayed in the Multiframe menus. In addition, the function of some of the Multiframe menu items change in order to support the Report Window: The menu items with modified behaviour and the additional menu items are as follows:

- **Group Menu**
- **Design Menu**
- **Code Submenu**
- **Display Menu**
- **Efficiency Submenu**
- **Help Menu**

Group Menu

The Group menu provides commands for organising the members in the structural model into groups or assemblies. The entries in this menu relevant to design are list below.

Create Design Member

Group the selected members together to form a multi-member design member.

Remove Design Member

Delete or split the selected members from multi-member design member(s).

Design Menu

The Design menu provides commands for checking and optimising the members in your structure.

Code

See “Code Submenu”

Check

Check the selected members in the Frame window for their compliance with the current code. You may use the Check dialog to choose which design calculations should be carried out and which load cases should be checked.

Design

Select the lightest weight sections for the selected members in the Frame window that will satisfy the design criteria. You may use the Design dialog to choose which design calculations should be carried out and which load cases should be examined.

Bending

Specify the design parameters controlling bending checks. Enter the unbraced lengths for the selected members in the Frame window and specify any web stiffener spacing.

Tension

Specify the design parameters used for tension checks. Specify the area of any boltholes, which must be subtracted from the cross-sectional area of the section when doing design calculations.

Compression

Specify the design parameters controlling compression checks. Allows you to select the effective lengths and the unbraced lengths for the selected members in the Frame window.

Combined

Specify the design parameters controlling combined bending and compression checks.

Serviceability

The Serviceability command allows you to set design information regarding serviceability of the frame this is currently only used for the AS4100 and NZS3404 design codes.

Seismic

Specify the design parameters controlling seismic design checks. This is currently only used for the NZS3404 design code to specify the category of a member.

Design Details

This command allows you to set all of the design information for the members selected in the Frame window. As a short cut, you can double click on a member to bring up this design dialog for that member.

Steel Grade

Specify the grade of steel for the selected members in the Frame window. You can choose from a list of standard grades or enter custom values for the yield and ultimate tensile strength.

Constraints

Specify whether there are any constraints on the size of section, which may be chosen for the selected members. You can also specify if you require all of the selected members to be of the same section type.

Frame Type

Specify whether the current frame is able to sway or is braced against horizontal movement.

Allowable Stresses

This command allows you to specify the allowable stress increase for each load case on the structure. The allowable increase is entered as a factor (usually 1.33 or 1.5).

Capacity Factors

The Capacity Factors command allows you to modify the capacity factors for the frame. This is only used with limit state design codes.

Use Best Sections

Automatically replace the section type of each member with its lightest weight section as chosen using the design command.

Code Submenu

The code menu allows you to select the design code you wish to use for checking. The current code is indicated with a check mark beside the item. This determines which code is used when you do design calculations.

AS1250

Not currently implemented

AS 4100

Australian steel design code.

AS 4600

Australian/New Zealand steel design code.

NZS 3404

New Zealand steel design code

BS5 950

British steel design code.

CISC

Not currently implemented

Eurocode

Eurocode 3 steel design code.

AIJ

Current Japanese steel design code.

ASD

American ASD steel design code.

LRFD

American LRFD steel design code

User

Allows the user to set their own design criteria and checks.

Edit User Code

This command lets you edit the design calculations that will be used when you choose to check or design a frame using the User code. You can choose which checks should be performed and what calculations should be used for each check. You can type in your own equations

Display Menu

The Display menu lets you control what s displayed in each of the windows.

Data / Design Details

Display a table in the Data window of the design information for each of the members in the frame. The table includes steel grade, effective and unbraced lengths and limits on the size of the section for the member

Results / Member Efficiency

Display a table in the Results window of the computed efficiency for each of the members in the frame. The efficiency is the ratio of the design action or stress to the design strengths according to the current design code expressed as a percentage.

Efficiency

See “Efficiency Submenu”

Efficiency Submenu

The items in this menu may be used to control which type of efficiency diagram is displayed in the Plot window. The items listed in this menu change according to the current design code.

AS 4100 and NZS3404

The following items are available in the Efficiency submenu when using the Australian / International version of Multiframe Steel Codes.

Overall

Display the Overall efficiency as a colour on each member for the current load case in the Plot window.

Bending (Major Section)

Display the Major Bending/Major Section Bending efficiency as a colour on each member for the current load case in the Plot window.

Bending (Major Member)

Display the Major Member Bending efficiency as a colour on each member for the current load case in the Plot window.

Bending (Major Shear)

Display the Major Shear efficiency as a colour on each member for the current load case in the Plot window.

Bending (Minor Section)

Display the Minor Bending/ Minor Section Bending efficiency as a colour on each member for the current load case in the Plot window.

Bending (Minor Shear)

Display the Minor Shear/Bending (Minor Shear) efficiency as a colour on each member for the current load case in the Plot window.

Tension

Display the Tension efficiency as a colour on each member for the current load case in the Plot window.

Compression (Section)

Display the Compression/Section Compression efficiency as a colour on each member for the current load case in the Plot window.

Compression (Major Member)

Display the Major Member Compression efficiency as a colour on each member for the current load case in the Plot window.

Compression (Minor Member)

Display the Minor Member Compression efficiency as a colour on each member for the current load case in the Plot window.

Combined (Major Section)

Display the Combined (Major Section) efficiency as a colour on each member for the current load case in the Plot window.

Combined (Minor Section)

Display the Combined (Minor Section) efficiency as a colour on each member for the current load case in the Plot window.

Combined (Major In-Plane)

Display the Combined (Major In-Plane) efficiency as a colour on each member for the current load case in the Plot window.

Combined (Minor In-Plane)

Display the Combined (Minor In-Plane) efficiency as a colour on each member for the current load case in the Plot window.

Combined (Out-of-plane)

Display the Combined (Out-of-plane) efficiency as a colour on each member for the current load case in the Plot window.

Combined (Biaxial Section)

Display the Combined (Biaxial Section) efficiency as a colour on each member for the current load case in the Plot window.

Combined (Biaxial Member)

Display the Combined (Biaxial Member) efficiency as a colour on each member for the current load case in the Plot window.

Primary Deflection

Display the Primary Deflection efficiency as a colour on each member for the current load case in the Plot window.

Secondary Deflection

Display the Secondary Deflection efficiency as a colour on each member for the current load case in the Plot window.

ASD / AIJ

The following items are available in the Efficiency submenu when using USA and Japan versions of Multiframe Steel Codes.

Overall

Display the Overall efficiency as a colour on each member for the current load case in the Plot window.

Major Bending

Display the Major Bending/Major Section Bending efficiency as a colour on each member for the current load case in the Plot window.

Minor Shear

Display the Minor Shear/Bending (Minor Shear) efficiency as a colour on each member for the current load case in the Plot window.

Major Deflection

Display the Major Deflection efficiency as a colour on each member for the current load case in the Plot window.

Minor Bending

Display the Minor Bending/ Minor Section Bending efficiency as a colour on each member for the current load case in the Plot window.

Minor Shear

Display the Minor Shear/Bending (Minor Shear) efficiency as a colour on each member for the current load case in the Plot window.

Minor Deflection

Display the Minor Bending/ Minor Section Bending efficiency as a colour on each member for the current load case in the Plot window.

Tension

Display the Tension efficiency as a colour on each member for the current load case in the Plot window.

Slenderness

Display the Slenderness efficiency as a colour on each member for the current load case in the Plot window.

Compression

Display the Compression/Section Compression efficiency as a colour on each member for the current load case in the Plot window.

Bending Tension

Display the combined Bending Tension efficiency as a colour on each member for the current load case in the Plot window.

Bending Compression

Display the combined Bending Compression efficiency as a colour on each member for the current load case in the Plot window.

Sway

Display the Sway efficiency as a colour on each member for the current load case in the Plot window.

Help Menu

Provides access to an on-line help system.

Multiframe Steel Codes Help

This command allows you to launch the table of contents of the Multiframe Steel Codes help file.

References

You may find the following books useful to refer to if you need information on the methods used to check members in Multiframe Steel Codes.

Manual of Steel Construction, Allowable Stress Design
American Institute of Steel Construction, New York, 1989, 9th Edition

Manual of Steel Construction, Load & Resistance Factor Design
American Institute of Steel Construction, New York, 1986, 1st Edition

Steel Buildings, Analysis and Design
S W Crawley & R M Dillon, John Wiley & Sons, New York, 1984, 3rd Edition

Structural Steel Design, LRFD Fundamentals
J C Smith, John Wiley & Sons, New York, 1988, 1st Edition

The Behaviour and Design of Steel Structures
N S Trahair and M A Bradford, Chapman and Hall, London, 1988

Australian Standard AS4100-1990: Steel Structures
Standards Australia

Australian/New Zealand Standard AS/NZS 4600-2005: Cold-formed Steel Structures
Standards Australian and New Zealand

Design of Cold-formed Steel Structures (to Australian/New Zealand Standard AS/NZS 4600:1996)
G. J. Handcock, Australian Institute of Steel Construction, Sydney, 1998, 3rd Edition

New Zealand Standard NZS 3404-1997: Steel Structures
Standards New Zealand

Multiframe Steel Codess Handbook
B.Gorenc, R. Tinyou and A. Syam, UNSW Press, Sydney, 1996, 6th Edition

Design Capacity Tables for Structural Steel. Volume 1: Open Sections
Australian Institute of Steel Construction, Sydney, 1994, 2nd Edition

Design Capacity Tables for Structural Steel Hollow Sections
Australian Institute of Steel Construction, Sydney, 1992, 1st Edition

Index

A

About this manual, 1
Acceptance Ratio, 20
AIJ, 33, 134
Allowable Stresses, 19, 20, 134
area reduction coefficient, 29
AS 4100, 134, 135
AS/NZS 4600, 86
AS/NZS4600, 77, 89
AS1250 to User, 134
AS4600, 77, 89
ASD, 134
ASD / AIJ, 137

B

Bending, 13, 27, 35, 62, 79, 91, 133
Bending (Major Member), 136
Bending (Major Section), 136
Bending (Major Shear), 136
Bending (Minor Section), 136
Bending (Minor Shear), 136
Bending Checks, 4
bending coefficient, 28
Bending Compression, 138
Bending Tension, 138
bolt holes, 29
BS5 950, 134

C

Capacity Factors, 134
Check, 132
Checking a Frame, 20
CISC, 134
Code Checks, 86, 98
Code Menu, 134
Column Restraints, 69
Combined, 133
Combined (Biaxial Member), 137
Combined (Biaxial Section), 137
Combined (Major In-Plane), 137
Combined (Major Section), 136
Combined (Minor In-Plane), 137
Combined (Minor Section), 136
Combined (Out-of-plane), 137
Combined Actions, 13, 31, 42, 71, 84, 96
Combined Checks, 5
compression, 30, 41, 69, 107, 121
Compression, 13, 29, 41, 68, 83, 95, 133, 138
Compression (Major Member), 136
Compression (Minor Member), 136
Compression (Section), 136
Compression Checks, 5
Constraints, 133
Coordinate Systems, 9
Create Design Member, 132
critical buckling load, 83, 95, 107, 120

D

Data, 135
Data Window, 6, 131
Design, 132
Design Checking Procedure, 86, 98
Design Constraints, 27
Design Constraints, 18
Design Details, 133, 135
Design Members, 4
Design Members, 7
Design Members, 12
Design Menu, 132
Design Properties, 13, 84, 96
Designing a Frame, 23
Display Menu, 135

E

Edit User Code, 135
effective length, 83, 95, 107, 120
  factor, 83, 95, 107, 120
Efficiency, 22, 135
Efficiency Menu, 135
Enabling Steel Designer, 4
Eurocode, 134

F

Finding Design Values, 25
Frame Type, 19, 133
Frame Window, 6, 131
Fu, 15, 17
Fy, 15, 17

G

Governing Load Cases, 22
Group Menu, 132

H

Help Menu, 138

K

Kx, 30, 42, 70, 108, 121
Ky, 30, 42, 70, 108, 121

L

Lbx, 27
Lby, 27
Lcx, 41
Load Window, 131
LRFD, 135

M

Major Bending, 137
Major Deflection, 138
Member Efficiency, 135
Menus, 132
Minor Bending, 138
Minor Deflection, 138
Minor Shear, 137, 138

N

NZS 3404, 134

O

Optimization, 25
Optimum Sections, 24
Overall, 135, 137

P

Plot Window, 6, 131
Primary Deflection, 137
Printing, 25

R

Remove Design Member, 132
Report Window, 7, 132
restraint, 83, 95, 107, 120
Result Window, 6, 131
Results, 135

S

Saving the report, 26
Saving your Work, 26
Secondary Deflection, 137
Section Constraints, 18
Section Type, 15
Seismic, 133
Seismic, 35, 43
Seismic Checks, 5
Serviceability, 42, 72, 133
Serviceability Checks, 5
Set Best Section, 25
Setting Properties, 12, 31, 44, 72, 77, 89
Shear Area, 10
Slenderness, 138
Starting Steel Designer, 4
Steel Designer Help, 139
Steel Grade, 15, 85, 97, 133
Sway, 138

T

Tension, 13, 28, 39, 66, 82, 94, 133, 136, 138
Tension Checks, 4

U

ultimate tensile strength, 15
Unbraced Length, 27, 30, 41, 83, 95
Unbraced Lengths, 69
Use Best Sections, 24, 134
User, 135

Y

yield strength, 15